PyroSim

A MODEL CONSTRUCTION TOOL FOR FIRE DYNAMICS SIMULATOR (FDS)

PyroSim User Manual

THUNDERHEAD ENGINEERING

2013

PyroSim User Manual

Table of Contents

PyroSim User Manual ..... **ii**
Table of Contents ..... **iv**
Figures ..... **viii**
Disclaimer ..... **xi**
Acknowledgements ..... **xii**
Chapter 1. Getting Started **1**
Introduction 1
Download and Install 1
Using a Different FDS Executable 2
Purchase PyroSim 2
Installing a Floating License 3
Additional FDS and Smokeview Documentation 7
System Requirements 7
Contact Us 7
Chapter 2. PyroSim Basics **8**
PyroSim Interface 8
Navigation View 8
3D View 9
2D View 13
Snapshots of Display 13
Preferences 13
Units 16
Color Schemes 17
Chapter 3. Working with Files **18**
Creating a New PyroSim Model 18
Saving a PyroSim Model 18
Open a Saved PyroSim Model 18
Preventing Changes to a Model 18
Importing FDS Models 19
Exporting FDS Models 19
Importing CAD Files 20
Chapter 4. Meshes **24**
Working with Meshes 24
Uniform Meshes 24
Nonuniform Meshes 25
Using Multiple Meshes 26
Additional Mesh Actions 29
Chapter 5. Materials **30**
Solid Materials 30
Liquid Fuels 31
Chapter 6. Surfaces **33**
Reserved Surfaces 33
Surface Types 34
Adding Textures to Surfaces 40
Chapter 7. Geometry (Basic Concepts) **41**

Obstructions 41
Holes 44
Vents 45
Groups 46
Floors 48
Adding a Background Image to a Floor 50
Chapter 8. Drawing in PyroSim 51
Drawing/Editing Tool Overview 51
Snapping 53
Precise Keyboard Entry 56
2D versus 3D Drawing 57
Obstruction Drawing Tools 60
Hole Drawing Tools 65
Vent Tool 65
Solution Mesh Tool 66
Mesh Splitter Tool 67
Device Tool 68
Planar Slice Tool 69
HVAC Node Tool 70
HVAC Duct Tool 71
Other Drawing Tools 71
Editing Objects 72
Transforming Objects 73
Painting Obstructions and Vents 76
Measuring Length/Distance 77
Chapter 9. Creating Complex Geometry 79
Curved Walls 79
Trusses 82
Roofs 83
Stairs 84
Chapter 10. Working with Geometry Objects 86
Selection 86
Context Menus 86
Undo/Redo 86
Copy/Paste 86
Double-Click to Edit 87
Translate and Copy Dialog 87
Mirror and Copy Dialog 87
Scale and Copy Dialog 88
Rotate and Copy Dialog 89
Object Visibility 90
Chapter 11. Species 91
Primitive Species 91
Lumped Species 91
Chapter 12. Reactions 93
Mixture Fraction Combustion 93
Custom Smoke 95
Chapter 13. Particles 97

Massless Tracers 97
Liquid Droplets 97
Fuel Droplets 99
Solid Particles 99
Activation 99
Global Parameters 99
Particle Clouds 99
Chapter 14. Devices 101
Aspiration Detection System 101
Gas or Solid Phase Device 102
Thermocouple 103
Flow Measurement 103
Heat Release Rate Device 103
Layer Zoning Device 104
Path Obscuration (Beam Detector) Device 104
Heat Detector 105
Smoke Detector 105
Sprinkler 105
Nozzle 106
Chapter 15. HVAC Systems 107
HVAC Duct 107
HVAC Node 108
HVAC Fan 109
HVAC Filter 109
HVAC Aircoil 110
HVAC Vents 111
Control Logic 112
Creating Activation Controls 112
Time-based Input 114
Detector-based Input 114
Chapter 16. Output Controls 116
Solid Profiles 116
Slices 116
Boundary Quantities 117
Isosurfaces 118
Plot3D Data 119
Statistics 120
Chapter 17. Evac 123
Using FDS+EVAC 123
Where to Find FDS+EVAC Records in PyroSim 124
Chapter 18. Running the Simulation 125
Simulation Parameters 125
Single-thread Execution 131
Parallel Execution 134
Cluster Execution 135
Resuming a Simulation 136
Chapter 19. Post-Processing 137
Launching Smokeview 137

Time History Results 137
Archiving Results 138
Restoring Archived Results 138
Chapter 20. Managing Data Libraries 140
Create and Manage Your Own Libraries 140
Use the Library Provided with PyroSim 140
Import a Material or Reaction from the FDS 4 Database 141
Chapter 21. Advanced FDS Parameters 142
Additional Records Section 142
Advanced Parameters 143
Chapter 22. Troubleshooting 144
Licensing/Registration Problems 144
Video Display Problems 144
Memory for Large Models 144
MPI Problems 144
Contacting Technical Support 146
Appendix A. Opening FDS v4 and PyroSim v2006 Files 147
Global Simulation Parameters 148
Sprinklers and Pipes 149
Reactions 149
Surfaces 149
Thermally Thin Surfaces 150
Where is the Surface Database? 150
Appendix B. Opening FDS v5 and PyroSim v2012 Files 151
Global Simulation Parameters 152
Reactions 152
Surfaces 153
Particles 153
References 155

Figures

Figure 1.1. Licensing and Activation Dialog 3
Figure 1.2. Display of Host Name and Host ID 4
Figure 1.3. Select License Server in dialog 6
Figure 1.4. Input Server and Port 6
Figure 2.1. Using the context menu in the Navigation View 9
Figure 2.2. 3D navigation toolbar 9
Figure 2.3. Exterior view of model (PyroSim model by John McKinney) 10
Figure 2.4. Interior view of model looking at roof and bleachers 11
Figure 2.5. Floors drop-down 12
Figure 2.6. Filter toolbar 12
Figure 2.7. Mesh filter toolbar 12
Figure 2.8. Filtering mesh elements 13
Figure 2.9. PyroSim Preferences 14
Figure 2.10. The materials dialog showing the FDS record preview pane 15
Figure 2.11. FDS Preferences 15
Figure 2.12. Display Preferences 16
Figure 3.1. DXF/DWG Import Options dialog 20
Figure 3.2. STL Import Options dialog 22
Figure 4.1. Defining properties of the new mesh 25
Figure 4.2. Defining properties of the nonuniform mesh 26
Figure 4.3. 3D display of first and second mesh 26
Figure 4.4. Correct and incorrect mesh alignment 29
Figure 6.1. The Edit Surfaces dialog 33
Figure 6.2. Effect of normal axis on the direction of tangential velocity 36
Figure 7.1. Conversion of a slab obstruction to FDS blocks 42
Figure 7.2. Obstruction dialog 42
Figure 7.3. A slab obstruction with a hole cut from it 44
Figure 7.4. Hole Properties dialog 45
Figure 7.5. Vents on a mesh boundary and obstruction 45
Figure 7.6. New Vent dialog 46
Figure 7.7. Create Group dialog 47
Figure 7.8. Dragging objects to a new group in the Navigation View 47
Figure 7.9. The Change Group dialog 48
Figure 7.10. The Group drop-down 48
Figure 7.11. Manage Floors dialog 48
Figure 7.12. New Floor dialog 49
Figure 7.13. Display of background image 50
Figure 8.1. Drawing/Editing toolbar 51

Figure 8.2. Pinning a drawing/editing tool 52
Figure 8.3. The quick action menu for the wall tool 53
Figure 8.4. Snap indicator 53
Figure 8.5. Sketch grid 54
Figure 8.6. Polar constraint at an angle of 45 degrees 55
Figure 8.7. Locked constraint 56
Figure 8.8. Tool editor window 57
Figure 8.9. Precise keyboard entry 57
Figure 8.10. Slabs in different planes aligned in the 2D View 58
Figure 8.11. Vent and devices attached to a wall 59
Figure 8.12. Objects stacked in the 3D View 59
Figure 8.13. Improper vs proper hole drawing in the 3D view 60
Figure 8.14. Snapping to another plane in the 3D View 60
Figure 8.15. Tool properties dialog for obstructions 61
Figure 8.16. A polygonal slab obstruction 62
Figure 8.17. A wall obstruction 63
Figure 8.18. Wall alignment options 63
Figure 8.19. Blocks drawn with the block obstruction tool 64
Figure 8.20. A room drawn with the room tool 65
Figure 8.21. A solution mesh drawn with the mesh tool 66
Figure 8.22. Two types of drawn meshes 67
Figure 8.23. Meshes being split by the mesh splitter tool 68
Figure 8.24. A gas-phase device being drawn in the 3D View 69
Figure 8.25. Drawing a slice with the planar slice tool 70
Figure 8.26. Using the HVAC Node drawing tool 70
Figure 8.27. Drawing an HVAC Duct 71
Figure 8.28. Editing handles on an object 72
Figure 8.29. A highlighted face handle 73
Figure 8.30. Moving an object using the Move Tool 74
Figure 8.31. Rotating an object with the Rotate Tool 75
Figure 8.32. Mirroring an object using the Mirror Tool 76
Figure 8.33. An obstruction face highlighted by the paint tool 77
Figure 8.34. Information displayed by the measure tool 78
Figure 9.1. Background image used for all curved wall examples 79
Figure 9.2. Background image settings for curved wall examples 80
Figure 9.3. A curved wall drawn with three different segment lengths 80
Figure 9.4. A curved wall drawn using individual blocks 81
Figure 9.5. A curved wall drawn using the rotate technique 82
Figure 9.6. Trusses created using PyroSim tools 83
Figure 9.7. A roof created with the slab obstruction tool 84
Figure 9.8. A stairway created with the replicate tool 85
Figure 10.1. The translate dialog 87

Figure 10.2. The mirror dialog 88
Figure 10.3. The scale dialog being used to scale an object 89
Figure 10.4. The rotate dialog being used to rotate an object 90
Figure 12.1. Reaction equation 93
Figure 12.2. Fuel panel of the Edit Reactions dialog 94
Figure 12.3. Fire Suppression panel of the Edit Reactions dialog 95
Figure 12.4. Byproducts panel of the Edit Reactions dialog 95
Figure 13.1. Particle Cloud dialog 100
Figure 14.1. Creating an aspirator sampler 102
Figure 14.2. Creating a new sprinkler 106
Figure 0.1. The Activation Controls dialog 112
Figure 0.2. The object popup in the Activation Controls dialog 113
Figure 0.3. The Activation Controls dialog 114
Figure 16.1. An example of a slice plane shown in Smokeview 117
Figure 16.2. An example of a boundary quantity shown in Smokeview 118
Figure 16.3. An example of an isosurface shown in Smokeview 119
Figure 16.4. A velocity Plot3D data shown in Smokeview 120
Figure 16.5. An example of statistic output 121
Figure 18.1. Time tab of the simulation parameters dialog 125
Figure 18.2. Output tab of the simulation parameters dialog 126
Figure 18.3. Environment tab of the simulation parameters dialog 127
Figure 18.4. Init Region dialog 128
Figure 18.5. Angled Geometry tab of the simulation parameters dialog 129
Figure 18.6. Merging converted blocks 130
Figure 18.7. Effect of thickening on converted blocks 130
Figure 18.8. Misc. tab on the simulation parameters dialog 131
Figure 18.9. The FDS Toolbar 131
Figure 18.10. The FDS simulation dialog 132
Figure 18.11. Expand Boundary Holes dialog 133
Figure 18.12. Hole along a mesh boundary 133
Figure 18.13. Improperly cut hole along mesh boundary in FDS 134
Figure 18.14. Properly cut hole along mesh boundary in FDS 134
Figure 18.15. The Cluster FDS Parameters dialog 136
Figure 19.1. Time History Results 137
Figure 19.2. Archive FDS Results dialog 138
Figure 19.3. Restore Archived Results dialog 139
Figure 20.1. Creating a library of materials 140
Figure 21.1. Additional Records Section 142
Figure 21.2. Additional Fields 143
Figure A.1. Example import warnings 148

Disclaimer

Thunderhead Engineering makes no warranty, expressed or implied, to users of PyroSim, and accepts no responsibility for its use. Users of PyroSim assume sole responsibility under Federal law for determining the appropriateness of its use in any particular application, for any conclusions drawn from the results of its use, and for any actions taken or not taken as a result of analyses performed using these tools.

Users are warned that PyroSim is intended for use only by those competent in the fields of fluid dynamics, thermodynamics, combustion, and heat transfer, and is intended only to supplement the informed judgment of the qualified user. The software package is a computer model that may or may not have predictive capability when applied to a specific set of factual circumstances. Lack of accurate predictions by the model could lead to erroneous conclusions with regard to fire safety. All results should be evaluated by an informed user.

Throughout this document, the mention of computer hardware or commercial software does not constitute endorsement by Thunderhead Engineering, nor does it indicate that the products are necessarily those best suited for the intended purpose.

Acknowledgements

We thank Kevin McGrattan, Simo Hostikka, Jason Floyd, Bryan Klein, and Glenn Forney in the Building and Fire Research Laboratory at the National Institute of Standards and Technology and the VTT Technical Research Centre of Finland. They are the primary authors of the Fire Dynamics Simulator and Smokeview, without which PyroSim would not exist. They have been gracious in their responses to our many questions.

We would like to gratefully acknowledge the RJA Group for their collaboration with Thunderhead engineering in the development of PyroSim. Feedback and testing from the engineers at RJA has improved the usability and quality of PyroSim.

Development of PyroSim was originally supported by the National Science Foundation under Grants DMI-0232401 and DMI-0349759. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the authors and do not necessarily reflect the views of the National Science Foundation.

Chapter 1. Getting Started

Introduction

PyroSim is a graphical user interface for the Fire Dynamics Simulator (FDS). FDS models can predict smoke, temperature, carbon monoxide, and other substances during fires. The results of these simulations have been used to ensure the safety of buildings before construction, evaluate safety options of existing buildings, reconstruct fires for post-accident investigation, and assist in firefighter training.

FDS is a powerful fire simulator which was developed at the National Institute of Standards and Technology (NIST) (McGrattan, et al., 2013). FDS simulates fire scenarios using computational fluid dynamics (CFD) optimized for low-speed, thermally-driven flow. This approach is very flexible and can be applied to fires ranging from stove-tops to oil storage tanks. It can also model situations that do not include a fire, such as ventilation in buildings. FDS and the Smokeview visualization program are both closely integrated into PyroSim.

The PyroSim interface provides immediate input feedback and ensures the correct format for the FDS input file. You can work in either metric or English units and you can switch between the two at any time. In addition, PyroSim offers high-level 2D and 3D geometry creation features, such as diagonal walls, background images for sketching, object grouping, flexible display options, as well as copying and replication of obstructions (0). You can import DXF files that include either 3D faces or 2D lines that can be extruded to create 3D objects in PyroSim (Chapter 3).

The PyroSim interface, for which this manual is written, supports Version 5 of the Fire Dynamics Simulator. PyroSim 2006, which supports Version 4 of the Fire Dynamics Simulator, is still available upon request.

Download and Install

The current version of PyroSim is available on the web at http://www.pyrosim.com. Licensed users have access to a direct download page. Other users can download PyroSim by requesting a free trial. There is no difference between the trial version of PyroSim and the full version. Any restrictions imposed by the trial version are based on the license.

When installing PyroSim, the installer will either upgrade an existing version or install PyroSim to a new location. This behavior is based on the version. When installing a minor update (e.g. upgrading from PyroSim 2008.1 to PyroSim 2008.2), the older installation will be removed and replaced with the new version. When installing a major update (e.g. PyroSim 2008.1 to PyroSim 2010.2), the older version will not be modified and the newer version will be installed to a different folder. Versions of PyroSim after PyroSim 2006 include their major version number in the install folder to make the difference between installed versions clear.

Administrator privileges are required to install PyroSim. The installer needs to add processes to the operating system for license management and parallel FDS simulation.

Using a Different FDS Executable

Each PyroSim release comes bundled with FDS. A particular PyroSim release is designed and tested against the bundled version of FDS, but can be used to run any version of FDS. However, PyroSim will generate an input file based on the bundled version of FDS and it is important to understand differences in input format between the FDS versions before customizing PyroSim's FDS version.

To change the version of FDS used by PyroSim:

1. Download and install a new version of FDS from NIST.
2. On the **File** menu, click **Preferences...**.
3. Click on the appropriate **Folder** icon in the **FDS Execution** section to select the executable you want PyroSim to use.
4. Click **OK** to close the **Preferences** dialog.

Purchase PyroSim

To enable PyroSim beyond the 30 day trial, you must purchase a license. A full list of purchasing options can be found at the PyroSim web site.

Online Purchase

You can purchase PyroSim online with our secure order form at:

https://www.thunderheadeng.com/php/buy.php

The order form is encrypted and Thunderhead Engineering will not retain your credit card number. Once the online transaction has been processed you will receive an **Activation Key**. Follow the instructions below to activate your license.

If you need to contact us, please refer to the Contact Us section later in this chapter.

Phone Purchase

Simply call Thunderhead and we will be happy to help you with the purchase. For contact information, please refer to the Contact Us section later in this chapter.

License Activation

When you purchase a license, you will receive an alphanumeric key that can be used to activate the software. You must enter this key in the **Licensing and Activation** dialog. To activate your license using Online Activation:

1. Start PyroSim. If the installation of PyroSim is not currently licensed, the **Licensing and Activation** dialog will automatically appear and you can skip to step 3.
2. On the **Help** menu, click **License...**.
3. Select the **Online Activation** option.

4. Enter your **Registration Key** into the **Key** box shown in Figure 1.1.
5. Click the **Activate** button.

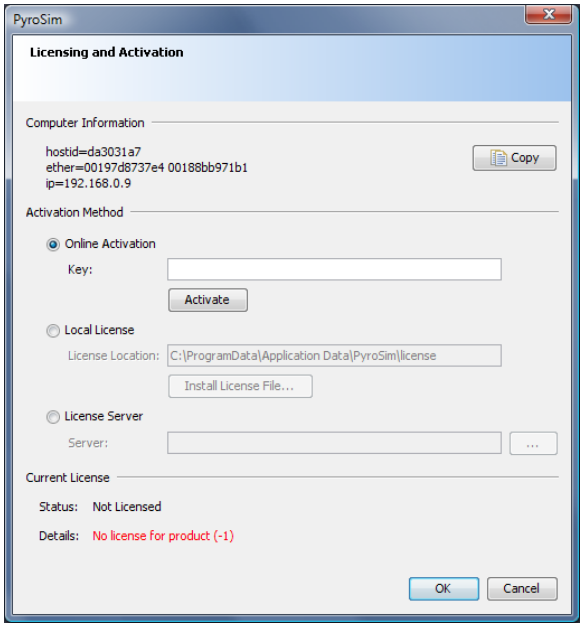


Figure 1.1. Licensing and Activation Dialog

Transferring a License

To transfer your license to another computer:

1. On the **Help** menu, click **Remove License…**.
2. Click **OK** to permanently remove your license.
3. A dialog indicating the license has been successfully removed will be displayed. Copy the **Confirmation code** and contact Thunderhead for a replacement license for the next computer.

PyroSim will exit when you close the **Remove License** dialog.

Updating PyroSim

To update your version of PyroSim, simply run the installer for the updated version. Your application files will be replaced and your license will remain intact.

If you purchased a perpetual license for PyroSim and your maintenance period has expired, new versions of PyroSim will not function with your old license and it will be necessary to reinstall the old version of PyroSim or purchase an updated license.

Installing a Floating License

A floating (network) license allows multiple users to run PyroSim from a central server. The PyroSim License Manager is required for floating (network) licenses of PyroSim. The license manager maintains a pool of licenses that can be checked out from copies of PyroSim installed anywhere on the network. The

license manager must be installed onto one computer that will act as a server for the floating licenses. Multiple PyroSim installations can then be configured to use the floating license server.

Install the Floating License Manager

The Floating License Manager should be installed on the computer that will act as the server for the floating licenses. To install the PyroSim License Manager:

1. Download the PyroSim License Manager installer (PyroSimLicenseMgr-setup.exe). This file can be downloaded from the PyroSim download page.
2. Log in using an account with administrative privileges.
3. Run the PyroSim License Manager installer.

This will install the main license server (rlm.exe), a PyroSim-specific license server (theng.exe), and utilities for managing the server. To enable the floating license server, you must now install a license file.

Install the License File

To activate your floating license, you must install a license file. This license file is generated by Thunderhead Engineering based on your server's Host Name and Host ID. To find your Host Name and Host ID:

1. On the **Windows Start Menu**, click **All Programs/PyroSim License Manager/Server HostID**. A command window will appear that displays the server's Host Name and Host ID, Figure 1.2.
2. Send your Host Name and Host ID to Thunderhead Engineering (email sales@thunderheadeng.com). Also, include your purchasing information to help us know what kind of license you need. Thunderhead Engineering will reply with a license file (pyrosim.lic).
3. Copy the license file (pyrosim.lic) to the license manager installation folder, typically C:\Program Files\PyroSim License Manager.
4. On the **Windows Start Menu**, click **All Programs/PyroSim License Manager/Restart License Service**. You must be an administrator to restart the license service. On Windows Vista, right-click the **Restart License Service** shortcut and select **Run as administrator**.

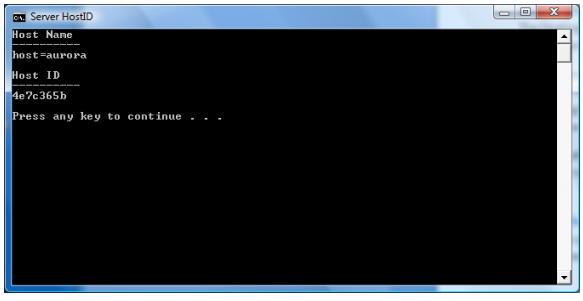


Figure 1.2. Display of Host Name and Host ID

Once the license service has been restarted, the license file will be loaded and the server is ready to provide PyroSim licenses to client computers.

Server Configuration

You can check the status of the license server using the Server Status shortcut, which displays a short textual report of server usage, or through the Monitor License Server shortcut, which will displays a simple web interface. Both methods show how many licenses are in use and which users are currently using licenses.

```
HOST aurora 4e7c365b 52100
ISV theng theng.exe theng.opt 52101
```

By default your license server uses TCP port 52100. If you are using a personal software firewall or accessing the server across a hardware firewall, you will have to configure the firewall to allow TCP traffic on ports 52100 and 52101. The ports used by the license manager can be configured in the license file by editing the port numbers on the HOST and ISV lines. After changing the port numbers in the license file, you must restart the server for the changes to take effect. Also, any PyroSim installations that were configured using the previous port assignments will need to be updated.

There are several advanced options available to administrators to control the behavior of the license manager. The PyroSim License Manager is based on the Reprise License Manager (RLM). Please see the RLM End-User Documentation for advanced server administration. The RLM documentation can be found in the PyroSim License Manager installation folder.

PyroSim Configuration on Local Computer

Configure PyroSim to locate the floating license server using the **Licensing and Activation** dialog.

To configure a PyroSim installation to use a floating license:

1. Start PyroSim. If the installation of PyroSim is not currently licensed, the **Licensing and Activation** dialog will automatically appear and you can skip to step 3.
2. On the **Help** menu, click **License...**
3. Under **Activation Method**, click **License Server**.
4. Click the **...** button to the right of the **Server** box to edit the server name and port number.
5. In the **Server** box, type your Host Name.
6. In the **Port** box, type 52100 (or the number you specified in the license file).
7. Click **OK** to save the server connection settings and attempt to connect to the license server.

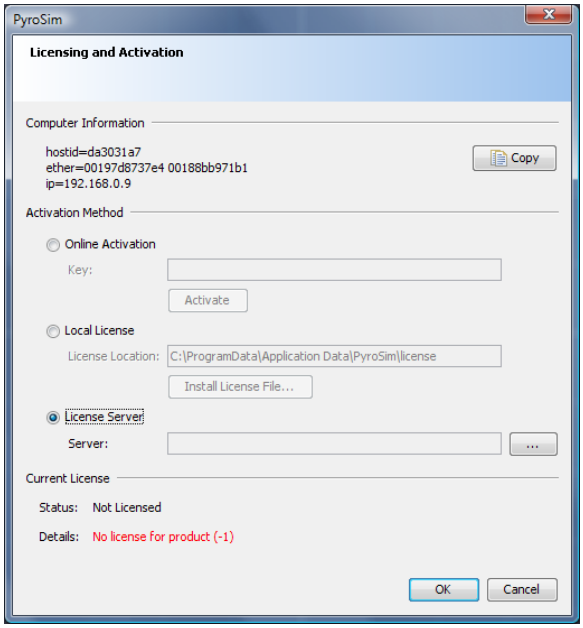


Figure 1.3. Select License Server in dialog

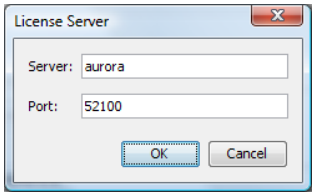


Figure 1.4. Input Server and Port

This completes the floating license configuration. This client will now check licenses in and out as needed.

Note

The default port number is 52100. If your configuration uses a custom port number on the HOST line of the server license file, use that port number instead.

The license server location can also be set for the client using the pyrosim.props configuration file. Set the PyroSim.licenseServer property to port@host, where port is the port number of the license manager, and host is the server name. Using the server name "aurora" from our previous example, this line would be:

```
PyroSim.licenseServer=52100@aurora
```

Property names in the pyrosim.props file are case sensitive, so the property name must be entered exactly as shown.

Additional FDS and Smokeview Documentation

In preparing this manual, we have liberally used descriptions from the FDS User's Guide (McGrattan, et al., 2013). The FDS Users Guide, the FDS Technical Reference, and the Smokeview Users Guide have been included with PyroSim. Updated documentation and executables for FDS and Smokeview may be available at: http://fire.nist.gov/fds/.

System Requirements

PyroSim runs on the Microsoft Windows operating system. You should have at least 256 MB of system RAM, and a graphics card that supports OpenGL 1.1 or later. If you want to perform realistic simulations, we recommend 2 GB of system RAM, a graphics card with at least 128 MB of graphics memory, and a dual or quad core. PyroSim transparently supports parallel processing on multi-core/multi-processor computers.

Contact Us

Thunderhead Engineering
403 Poyntz Avenue, Suite B
Manhattan, KS 66502-6081
USA

Sales Information: sales@thunderheadeng.com
Product Support: support@thunderheadeng.com
Phone: +1.785.770.8511

Chapter 2. PyroSim Basics

PyroSim Interface

PyroSim provides four editors for your fire model: the 3D View, 2D View, Navigation View, and the Record View. These all represent your current model. If an object is added, removed, or selected in one view, the other views will simultaneously reflect the change. Each view is briefly described below.

- **Navigation View**: This view lists many of the important records in the model. It allows you to organize your model geometry into groups such as room or sofa. Locating and modifying records is often faster and easier in this view.
- **3D View**: This view shows a 3D representation of your current fire model. You can explore the model using different view controls. You can also control the appearance of the model with options like smooth shading, textures, and object outlines. Geometric features can also be changed.
- **2D View**: This view is useful for quickly sketching geometry such as walls and furniture. You can choose from three viewing planes and perform many useful geometric manipulations.
- **Record View**: This view gives a preview of the FDS input file that will be generated for the simulation. It also provides a way to add custom records that will not be processed by PyroSim, but will be sent to FDS.

Navigation View

The navigation view is a tree-like view on the left side of the PyroSim main window. An example of this view in use is shown in Figure 2.1. When you right-click on an item in this view, a list of the functions PyroSim can perform on that item is shown. To rearrange objects in the Navigation view, make a selection and then drag the object(s) to the new location.

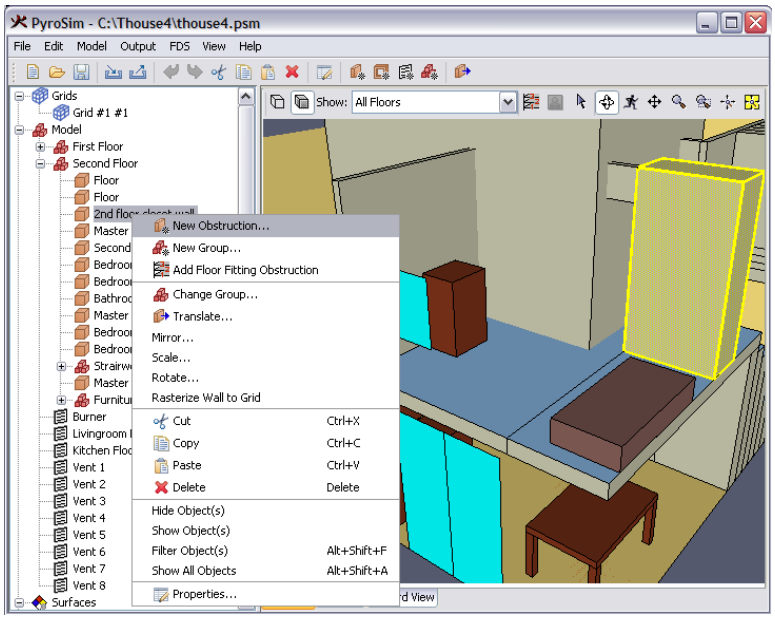


Figure 2.1. Using the context menu in the Navigation View

3D View

Use the 3D view to rapidly obtain a visual image of the model and perform some drafting. In this view, the user can navigate through the model in 3D and select objects. This view also provides display filters to quickly show/hide entire categories of objects or switch between floors. In addition, any drafting that requires objects to be snapped to faces of other objects, such as drawing a vent on an obstruction or attaching a measuring device to a solid can be easily achieved in this view. For more information on drafting, see Chapter 8.

Navigation/Selection

There are several tools that can be used to navigate the model and select objects. The tools for the 3D view are found in the **navigation toolbar** above the 3D view as shown in Figure 2.2.

Figure 2.2. 3D navigation toolbar

- **Select/Manipulate Tool ( ):** This general-purpose tool can be used to perform most navigation activities.
    - **Selection:** Left-click an object to select it. Drag the left mouse button to draw a selection box and select all objects within the box. Double-click the left mouse button to select and open a properties dialog for the object under the cursor. Hold **ALT** while performing selection to select the hierarchical parent of the object under the cursor. Right-click to show a context menu for selected items under the cursor.
    - **Panning:** Drag the middle mouse button to **pan** the model.
    - **Orbiting:** Drag the right mouse button to **orbit** the model.

- **Manipulating:** If a single object is selected, it may show manipulation handles (blue dots or faces). Left-click one of the handles to begin manipulation or drag the left mouse button to perform manipulation in one gesture. For more information on manipulation, see Editing Objects on page 72.

- **Orbit Tool ( ):** This is another general-purpose tool that may be more familiar to existing PyroSim users, but it is more limited than the **Select/Manipulate Tool**.
  - **Selection:** This is the same as with the **Select/Manipulate Tool** except that a selection box cannot be drawn.
  - **Orbiting:** Drag any mouse button to **orbit**.
  - **Panning:** Hold the **SHIFT** key down while dragging any mouse button to **pan**.
  - **Zooming:** Hold the **ALT** key down while dragging any mouse button to **zoom**.
  - **Manipulating:** This tool cannot perform manipulation.
- **Roam Tool ( ):** This tool allows the user to move into the model rather than viewing it only from the outside as shown in Figure 2.4. This tool can take some experimentation, but once mastered, it can provide unique views of the model.
  - **Selection:** This is the same as with the **Orbit Tool**.
  - **Looking:** Drag any mouse button to look around. This pivots the camera about the camera's location, similar to a first-person video game.
  - **Moving:** Hold the **ALT** key while dragging a mouse button to move the camera up and down along the Z axis. Hold the **CTRL** key while dragging a mouse button to move the camera forward, backward, and side-to-side in the camera's XY plane.
- **Pan Tool ( ), Zoom Tool ( ), Zoom Box Tool ( ):** These tools break out the functionality of the above tools so that dragging any mouse button will perform the needed action.

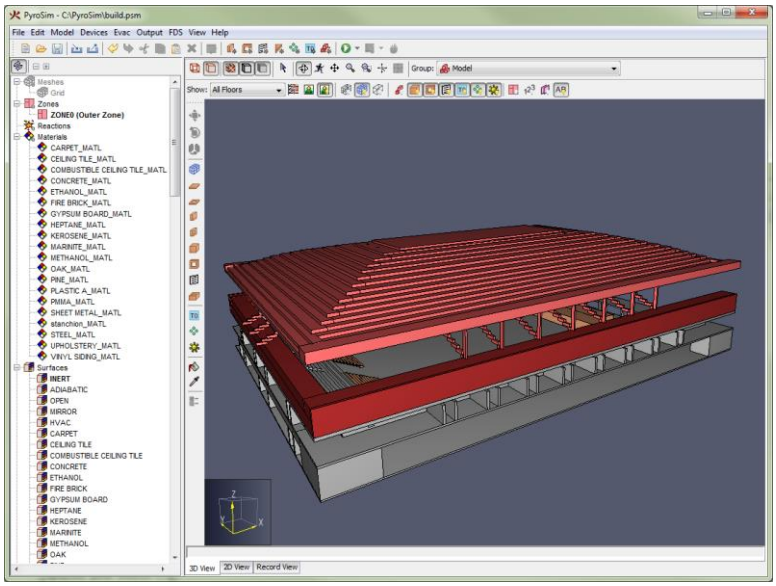


Figure 2.3. Exterior view of model (PyroSim model by John McKinney)

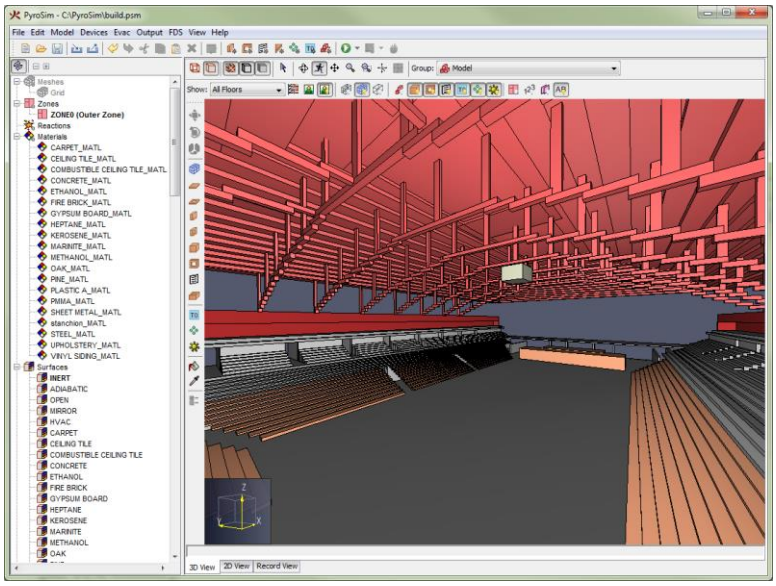


Figure 2.4. Interior view of model looking at roof and bleachers

Zooming

The model can also be zoomed in and out with any of the navigation tools by using the scroll wheel. Scrolling up zooms in and scrolling down zooms out. With all but the **Roam Tool**, using the scroll wheel will zoom in on the point under the cursor. With the **Roam Tool**, the scroll wheel only zooms the center of the view.

Resetting the View

At any time, the camera's view can be **reset** to see the entire extents of the model by clicking the **Reset View** button ( ) or pressing **CTRL+r**. In addition, the camera can be reset to only the currently selected objects by clicking the **Reset to Visible** button ( ) or pressing **CTRL+e**. Resetting the view also has the effect of changing the orbit center when orbiting.

Orbiting

Orbiting is the action of spinning the camera about its focal point, which is the center of the model or center of the selection, depending on which **reset** action was last performed. By default, orbit works as if there is an invisible sphere around the model on which you click and drag the mouse to spin. Alternatively, orbiting can be performed similarly to Smokeview by going to the **View** menu and selecting, **Use Smokeview-like Navigation**. In this mode the camera spins about the Z axis with left and right mouse movements and about the local X axis with up and down movements.

Filtering

There are several ways to filter the objects shown in the **3D** view. Filtering can be performed with clipping planes that are associated with floors or through filter buttons that can quickly show/hide categories of objects.

To use clipping, the user must first define floors for the model as discussed in Floors on page 48. Once the floors are defined, a floor can be selected by using the **Floor Drop-down** above the **3D** or **2D** view as shown in Figure 2.5.

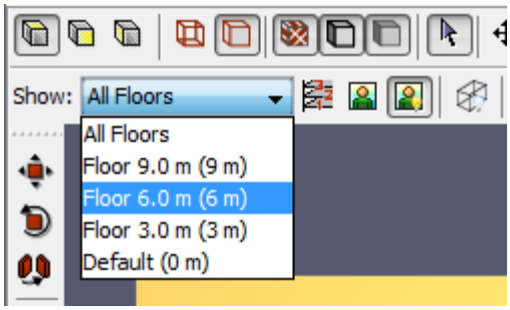


Figure 2.5. Floors drop-down

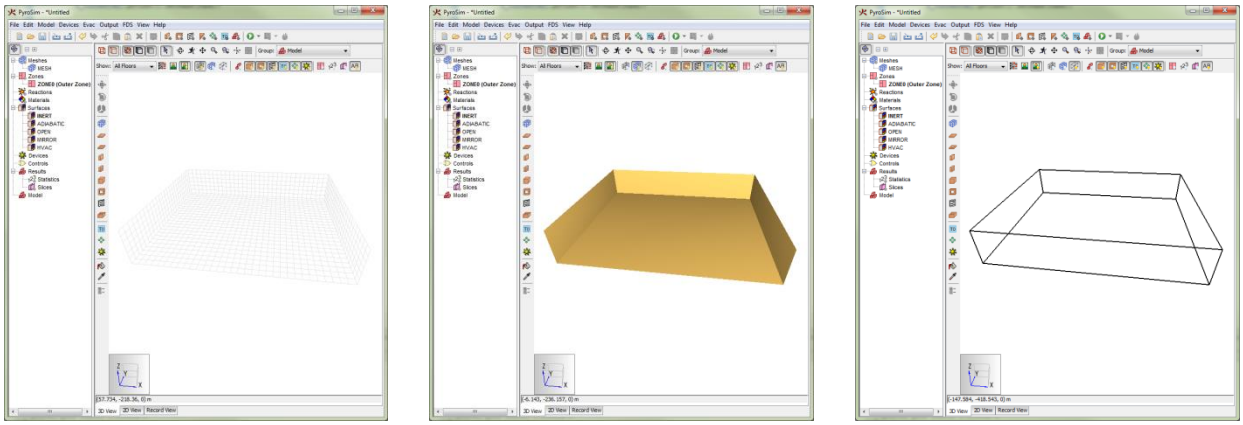
Once a floor has been selected, its clipping planes will be applied to the entire scene to only show objects within the clipping region.

Filtering can also be performed using the filter toolbar buttons as shown in Figure 2.6. Selecting/deselecting these buttons will quickly show/hide all objects of a specific type, such as obstructions, holes, vents, etc.

Figure 2.6. Filter toolbar

Filtering can also be applied to meshes but in a slightly different way. Instead of showing/hiding all meshes, the user can selectively show/hide three different elements of them using the mesh filter toolbar shown in Figure 2.7. This toolbar selectively allows viewing mesh grid lines, mesh boundaries, and mesh outlines. Figure 2.8 shows the different mesh elements. In the figure, A shows the grid lines, B shows the boundary, and C shows the outline.

Figure 2.7. Mesh filter toolbar



A B C

Figure 2.8. Filtering mesh elements

Background images attached to floors can be quickly shown/hidden using the **Show Background Image** filter button next to the floor drop-down.

2D View

The **2D** view is mostly the same as the **3D** view with some key differences:

- The 2D view provides pre-set, orthographic views of the model. The view can be set to the **top**, **front**, or **side** views.
- The 2D view does not have the **Orbit** and **Roam** tools.
- The **Select/Manipulate** tool **pans** the model when dragging the right mouse button rather than **orbiting**.
- Drafting is performed differently in the 2D View. For more information, see 2D versus 3D Drawing on page 57.
- Snapping can be performed to either the solution mesh or to a user-defined sketch grid.
- When snapping to a solution mesh, the mesh grid lines are always shown, the boundary is never shown, and only the outline of the mesh can be optionally shown.
- Floor clipping is slightly different. For more information see Floors on page 48.

Snapshots of Display

Images of the current display can be saved to a file by opening the **File** menu and clicking **Snapshot...**. The user can specify the file name, image type (png, jpg, tif, bmp), and the resolution. A good choice for image type is Portable Network Graphics (png).

Preferences

PyroSim preferences can be set by going to the **File** menu and choosing **Preferences....** Any changes to the preferences will be set for the current PyroSim session and be remembered the next time PyroSim is started. The preferences are split into several groups, including **PyroSim**, **FDS**, and **Display** preferences.

PyroSim Preferences

These describe global PyroSim preferences as shown in Figure 2.9.

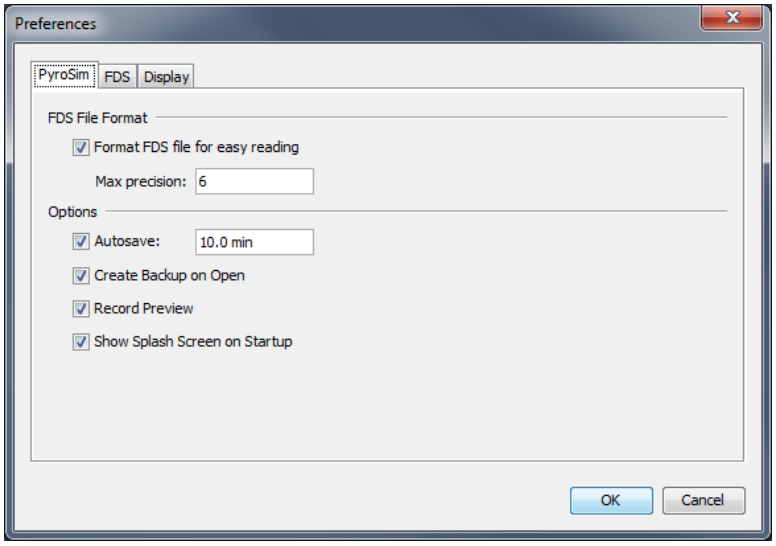


Figure 2.9. PyroSim Preferences

Format FDS file for easy reading - controls the format of real numbers in the FDS input file created by PyroSim. When this is checked, a real number is written such that if the absolute value is >= .001 and < 10000, it is written in decimal notation; otherwise, it is written in scientific notation. Max precision controls the number of significant digits written to the input file.

- **Autosave** - instructs PyroSim to periodically create a backup of the current model that is deleted when the PyroSim model is closed. This backup is useful in case PyroSim crashes or the computer loses power. The default setting enables this feature and saves every 10 minutes. In some cases, when working with large models, this can cause unexpected delays during the save and some users prefer to disable the feature and save manually.
- **Create Backup on Open** - controls whether PyroSim makes a backup of the PyroSim file after successfully opening it. This backup remains on disk when the file is closed, and so it can be used in case the main PyroSim file somehow becomes corrupted. NOTE: if this is enabled, PyroSim files may take slightly longer to load depending on how quickly the file can be copied to backup.
- **Record Preview** - adds a preview pane to many of the dialogs in PyroSim. This preview pane shows the text that will be produced for the FDS input file. This can be helpful for users that want to understand exactly how PyroSim options correspond to FDS input. An example can be seen in Figure 2.10.
- **Show Splash Screen on Startup** - controls whether the PyroSim splash screen is shown when starting PyroSim.

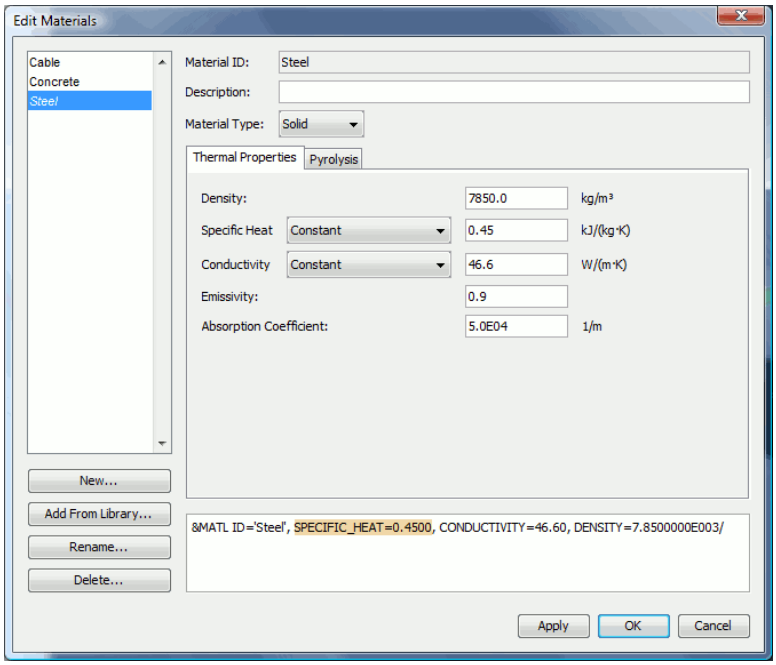


Figure 2.10. The materials dialog showing the FDS record preview pane

FDS Preferences

These preferences define the execution of FDS within PyroSim. They can be seen in Figure 2.11.

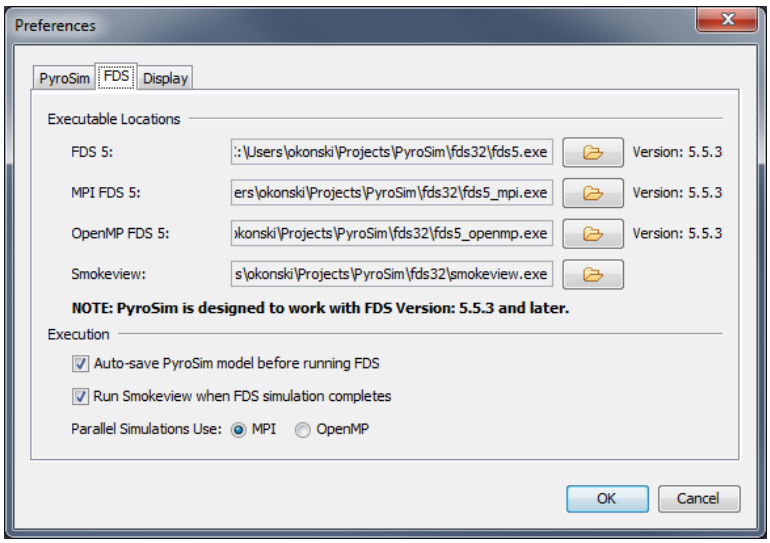


Figure 2.11. FDS Preferences

- **Executable Locations** - allow you to specify the FDS and Smokeview executables that are used by PyroSim. NOTE: PyroSim is designed to work with a specific version of FDS as noted below the executable locations. If you specify a different version, you may encounter unpredictable FDS results or errors.

- **Auto-save PyroSim model before running FDS** - controls whether PyroSim automatically saves the current PyroSim file just before beginning an FDS simulation.
- **Run Smokeview when FDS simulation completes** - indicates whether to automatically launch Smokeview when an FDS simulation completes.
- **Parallel Simulations Use:** - selects the FDS executable used when you click **Run Parallel...** on the **FDS** menu. The MPI option uses a separate process for each mesh in the simulation. The OpenMP option attempts to process loops in parallel and can improve performance on simulations that use one or many meshes.

Display Preferences

These preferences define advanced 2D and 3D display properties, as shown in Figure 2.12. They can be used to improve display performance on complex models, but they tend to create problems for some graphics cards, including crashing. For this reason, they are turned off when running in safe mode.

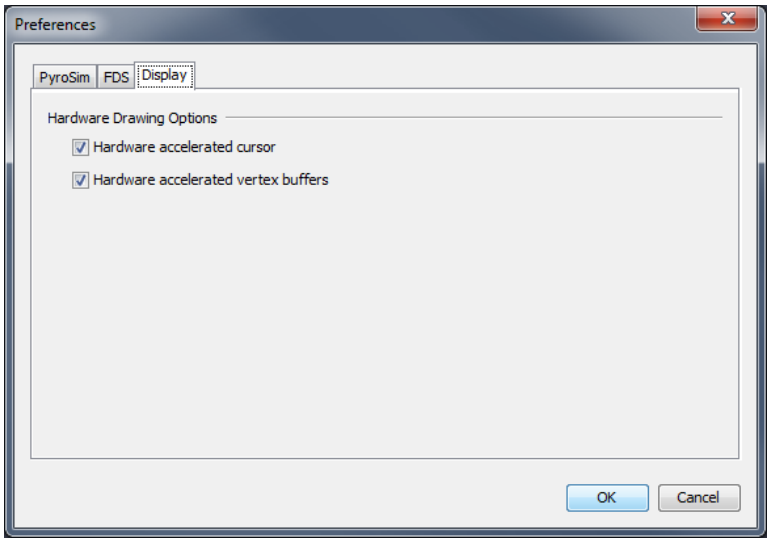


Figure 2.12. Display Preferences

- **Hardware accelerated cursor** - when checked, mouse crosshairs and object editing will remain smooth even on complex geometry that may be slow to pan or orbit. PyroSim does this by rendering the scene into an image buffer and then rendering that buffer as a texture underneath the crosshairs or editing geometry whenever the mouse cursor moves.
- **Hardware accelerated vertex buffers** - when checked, this can significantly improve rendering performance of complex geometry. PyroSim does this by storing the geometry in vertex buffers on the graphics card. PyroSim then makes very few OpenGL calls to render the geometry.

Units

Models can be created in either English or Metric units. To select a system of units, on the **View** menu, click **Units**, then click the desired unit. PyroSim will automatically convert your previous input values into the unit system you select. The **Record View** will always display values in the appropriate FDS units, regardless of what unit system you choose to work in.

Color Schemes

To select a Default, Black Background, White Background, or Custom color scheme, on the **View** menu, click **Color Scheme**. The custom color scheme is defined in the PyroSim.props file in the PyroSim installation directory (usually C:\Program Files\PyroSim).

1. To define a custom color scheme:
2. Close PyroSim
3. Edit the PyroSim.props file
4. Change the following default colors to the colors you wish:

```
Colors.Custom.axis=0xffff00
Colors.Custom.axis.box=0x404040
Colors.Custom.axis.text=0xffffff
Colors.Custom.background=0x0
Colors.Custom.boundary.line=0xffffff
Colors.Custom.grid=0x4d4d66
Colors.Custom.group.highlight=0xffff00
Colors.Custom.heatDetector=0xff0000
Colors.Custom.obst=0xff0000
Colors.Custom.obst.highlight=0xb2b200
Colors.Custom.origin2D=0x737373
Colors.Custom.smokeDetector=0xff00
Colors.Custom.snap.point=0xff00
Colors.Custom.snapto.grid=0x404040
Colors.Custom.snapto.points=0xc0c0c0
Colors.Custom.sprk=0xff
Colors.Custom.text=0xffffff
Colors.Custom.thcp=0xffff00
Colors.Custom.tool=0xff00
Colors.Custom.tool.guides=0x7c00
```

5. Save the edited PyroSim.props file
6. Restart PyroSim

Chapter 3. Working with Files

Several files are used when performing a fire analysis using PyroSim. These include the PyroSim model file, the FDS input file, and FDS output files. This section describes how to load and save files in the formats supported by PyroSim.

Creating a New PyroSim Model

When PyroSim is started, it begins with an empty model. You can close the current model and create a new empty model by opening the **File** menu and clicking **New**. PyroSim always has one (and only one) active model.

Saving a PyroSim Model

The PyroSim model file (PSM) is stored in a binary format that represents a PyroSim model. The PyroSim model contains all the information needed to write an FDS input file, as well as additional information such as obstruction grouping, floor heights, background images, and textures. This format is ideal for sharing your models with other PyroSim users.

To save a new model:

1. On the **File** menu, click **Save**.
2. Enter the file name and click the **Save** button.

Open a Saved PyroSim Model

PyroSim model files have a PSM extension. To open a saved model:

1. On the **File** menu, click **Open...**.
2. Select the file and click the **Open** button.

A list of recently opened files is also available. To open recent files, on the **File** menu, click **Recent PyroSim Files**, then click the desired file.

PyroSim has an auto-save feature which stores a copy of your current model every 10 minutes. This file is automatically deleted if PyroSim exits normally, but if PyroSim crashes, you can recover your work by opening the autosave file. It can be found either in the same directory as your most recent PSM file, or in the PyroSim installation directory if your model was unsaved.

For more information about opening files saved with previous versions of PyroSim, please refer to Chapter 22.Appendix A.

Preventing Changes to a Model

PyroSim supports write protection for a model. When write protection is enabled, users cannot modify a model (e.g. change geometry, edit surface properties, etc). This option can be enabled with or without password protection. If a model is write-protected, PyroSim will display notification in the application title bar.

To add write protection to a model:

1. On the **File** menu, click **Write Protection...**
2. Click **OK**.

The model will now be write-protected. Since a password was not used, a password will not be required to remove write protection.

To remove write protection from a model:

1. On the **File** menu, click **Write Protection...**
2. Click **OK**.

The model can now be edited. If needed, the dialog will require a password to release the lock.

Importing FDS Models

PyroSim allows you to import existing FDS input files. When you import an FDS file, PyroSim will create a new PyroSim model from the imported file. During import, PyroSim will check for the validity of each record. If errors are detected, you will be notified. You may then make the required corrections and attempt to import the file again.

To import existing FDS models into PyroSim:

1. On the **File** menu, select **Import FDS/CAD File…** or click the **Import** button on the main toolbar.
2. Select the FDS file and click **Open**.

PyroSim supports file import for versions 4, 5, and 6 of FDS. For more information about opening files compatible with version 4 or 5 of FDS, please refer to Appendix A.

Exporting FDS Models

PyroSim also allows you to explicitly export the current model to an FDS input file. You can manually edit the file to take advantage of advanced FDS features, or to easily transfer the input file to a different machine or special version of FDS.

To export an FDS file:

1. On the **File** menu, click **Export**, then click **FDS File...** or click the **Export** button on the main toolbar.
2. Enter the file name and click **Save**.

The file exported by PyroSim will be compatible with version 6 of FDS.

Importing CAD Files

PyroSim allows users to import several types of CAD file formats, including the DXF and DWG format and the STL (Standard Tessellation Language) format. Each type of file provides a variety of geometry that can either be directly represented as obstructions or as drawing guides in the PyroSim model.

Unlike FDS import which completely replaces the current PyroSim model, CAD import appends the data to the current model. This facilitates the ability to import data from several CAD files into one PyroSim model. This is useful when there is one blueprint per floor of a building or a 3D building has been split into several sections, each in a separate file.

DWG/DXF Import

DWG and DXF files are standard drawing formats supported by a variety of CAD software including Autodesk's AutoCAD and Revit software suites. These types of files can contain both 3D geometry, such as walls, slabs, solids, and faces and 2D geometry, such as circles, arcs, and lines.

While versions of PyroSim earlier than 2012 required users to decompose some types of geometry (such as solids or architectural elements) in the source software, PyroSim 2012 and later fully support these formats and all types of geometry without decomposition.

To import DXF or DWG files, perform the following:

1. On the **File** menu, select **Import FDS/CAD File…** or click the **Import** button on the main toolbar.
2. Select the desired DXF or DWG file to import and click **Open**.
3. Select the length unit from the **Import Options** dialog shown in Figure 3.1. If the DXF/DWG file stored a length unit with the file, this will be selected by default.
4. Click **OK** to begin import.

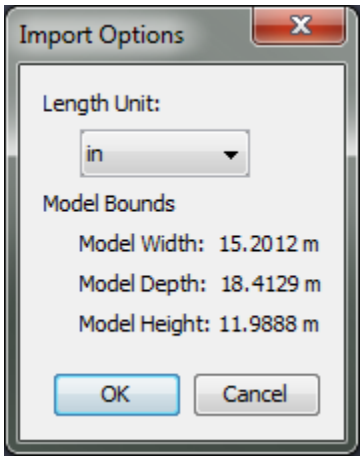


Figure 3.1. DXF/DWG Import Options dialog

In the **Import Options** dialog, the **Model Bounds** are shown to help the user choose the proper length unit if it is unknown. The **Model Bounds** will change depending on which unit is chosen. Depending on the size of the import file, the model bounds may not be shown until the user clicks **Calculate**. In this

case, it may take several minutes to calculate the model bounds. It may be more beneficial to simply import the model using the default unit or take a guess and then resize the model after import.

When PyroSim imports a DWG/DXF file, it will treat all 3D face data as obstructions and all other data (lines, curves, etc.) as separate CAD data. If an entity in the file contains both face and CAD data, the entity will be split into two entities so that CAD data can be easily deleted or hidden after import using the CAD filter button on the 3D/2D View toolbar (see Filtering on page 11).

An object with CAD data can be snapped to while drawing in PyroSim but is not converted to any type of FDS geometry.

An entity with face data will either be treated as a single, solid obstruction with some volume or as a collection of thin obstructions depending on the entity type in the DWG/DXF file. These objects will be represented as FDS geometry. The following entity types are treated as solids in PyroSim:

- 3D Solid
- Mass Element
- Mass Group
- Roof
- Slab1
- Roof Slab
- Stair
- Wall[1]
- Door
- Window
- Curtain Wall
- Curtain Wall Unit
- Curtain Wall Assembly
- Structural Member

All other entities containing faces, such as polygon meshes and polyface meshes are treated as collections of thin obstructions by PyroSim. They cannot be reliably treated as solid since there is no guarantee that their faces form a closed and non-self-intersecting shell or that this would even be desired.

Once the file is imported, PyroSim creates a hierarchy of groups and objects, such that there is one top group, named after the file. On the next level, there is a group for every layer containing geometry. Under each layer group there are one or more objects representing the entities in the file. The following illustrates the hierarchy as it would appear in the **Navigation View**:

[1] While PyroSim allows the user to draw walls and slabs, it will not convert DWG/DXF wall and slab entities to native walls and slabs, due to the complexity with which walls and slabs can be represented in the DWG/DXF file. They will instead be represented as collections of individual faces, representing one solid object.

- FileName
  - Layer1
    - Entity1
    - Entity2
    - …
  - Layer2
    - Entity3
    - Entity4
    - …
  - …

If the DWG/DXF file contains a block insert and the block contains entities from multiple layers, the block insert is split into several PyroSim objects, one for each layer of the block's originating entities. If all the entities in the block are from the same layer, however, there will be one resulting PyroSim object that will belong to the group corresponding to the block's entities' layer rather than the block insert's layer.

STL Import

PyroSim can also import objects from STL files, which are simply listings of triangles. Usually, each STL file represents the shell of one 3D solid object. To import an STL file, perform the following:

1. On the **File** menu, select **Import FDS/CAD File…** or click the **Import** button on the main toolbar.
2. Select the desired STL file and click **Open**.
3. Enter the import options in the **STL Import Options** dialog as shown in Figure 3.2.
4. Click **OK** to begin import.

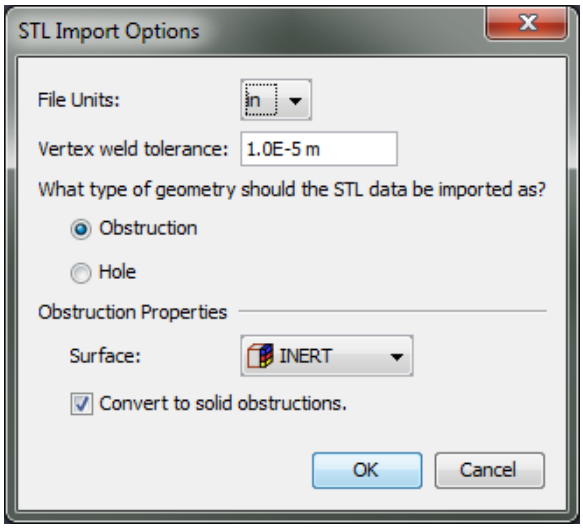


Figure 3.2. STL Import Options dialog

In the **STL Import Options** dialog, the following options can be specified:

- **File Units** – the units used to store the 3D coordinates in the STL file.
- **Vertex weld tolerance** – a distance used to determine how far apart vertices must be to be considered separate.
- **Resulting Geometry Type** – choose **Obstruction** to treat the resulting objects as obstructions and **Hole** to treat them as holes.
- **Surface** – the surface to apply to the resulting obstructions if applicable.
- **Convert to solid obstructions** – whether to treat the resulting objects as solid obstructions. If this is unchecked, each resulting object will be a collection of thin obstructions.

Because the STL file is simply a listing of triangles, there may be more than one object represented in the file. PyroSim will use the vertex weld tolerance to detect triangle connectivity and determine if there are several, disconnected sets of faces in the file. If there are, there will be one resulting PyroSim object per connected set of faces.

In addition, if the solid option is enabled or the objects are being treated as holes, import will only succeed if each face set is detected as a closed shell by PyroSim.

Working with Revit Architecture

While PyroSim cannot directly import Autodesk Revit files (RVT), Revit can export to DWG, which can then be imported into PyroSim. To export a DWG file in Revit Architecture 2012, perform the following:

1. Open the desired RVT file within Revit Architecture.
2. Click the Revit icon at the top left .
3. Select **Export->CAD Formats->DWG files**.
4. In the **DWG Export** dialog, click the **…** button in the **Select Export Setup** section.
5. In the **Modify DWG/DXF Export Setup** dialog, select the **Solids** tab, and select **ACIS solids**. This option allows PyroSim to import objects as solid obstructions rather than treating each exported face as a thin obstruction.
6. Click **OK** to close the **Modify DWG/DXF Export Setup** dialog.
7. In the **DWG Export** dialog, for **Export**, select `<In session view/sheet set>`.
8. For **Show in list**, select `Views in the Model`.
9. Click the **Check None** button, and then in the view table, select the check box for `3D View: {3D}` (Other views may be chosen, but the DWG will only contain entities visible in the selected views).
10. Click the **Next…** button and choose a file name for the DWG file.
11. Click **OK** to create the DWG.

The resulting DWG file can now be imported into PyroSim.

Chapter 4. Meshes

Working with Meshes

All FDS calculations are performed within computational meshes. Every object in the simulation (e.g. obstructions and vents) must conform to the mesh. When an object's location doesn't exactly conform to a mesh, the object is automatically repositioned during the simulation. Any object that extends beyond the boundary of the physical domain is cut off at the boundary. There is no penalty for defining objects outside of the domain, but these objects do not appear in Smokeview.

To achieve optimal simulation accuracy, it is important to use mesh cells that are approximately the same size in all three directions.

FDS uses a Poisson solver based on Fast Fourier Transforms (FFTs). A side effect of this approach is that optimal mesh divisions are constrained to the form $2^u\ 3^v\ 5^w$, where u, v and w are integers. For example, $64 = 2^6$, $72 = 2^3 * 3^2$, and $108 = 2^2 * 3^3$ are good mesh dimensions. However, 37, 99 and 109 are not. In addition, using a prime number of cells along an axis may cause undesirable results. PyroSim warns when the number of divisions is not optimal.

Uniform Meshes

This example illustrates creating a multiple mesh model. To create the first mesh:

1. On the **Model** menu, click **Edit Meshes...**.
2. Click **New**.
3. In the **Max X** box, type `5.0`, in the **Max Y** box, type `1.0`, and in the **Max Z** box, type `1.0`.
4. In the **X Cells** box, type `50`, in the **Y Cells** box, type `10`, and in the **Z Cells** box, type `10`.
5. Click **OK**.

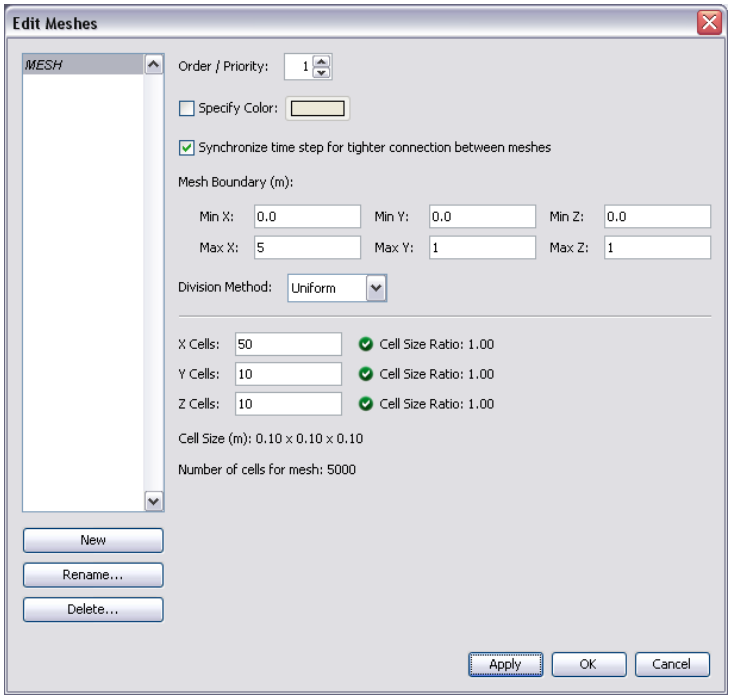


Figure 4.1. Defining properties of the new mesh

The 3D View will now display the resulting mesh.

Nonuniform Meshes

To create a second, nonuniform mesh:

1. On the **Model** menu, click **Edit Meshes...**.
2. Click **New**
3. In the **Min X** box, type `0.0`, in the **Min Y** box, type `1.0`, and in the **Min Z** box, type `0.0`
4. In the **Max X** box, type `1.0`, in the **Max Y** box, type `3.0`, and in the **Max Z** box, type `1.0`
5. In the Division method box, select Non-Uniform
6. In the table, enter the data shown in Table 4.1
7. Click **OK**

Table 4.1. Non-Uniform Mesh Parameters

Dir (X,Y,Z)	Num Cells	Size
X	10	0.1
Y	10	0.1
Y	5	0.2
Z	10	0.1

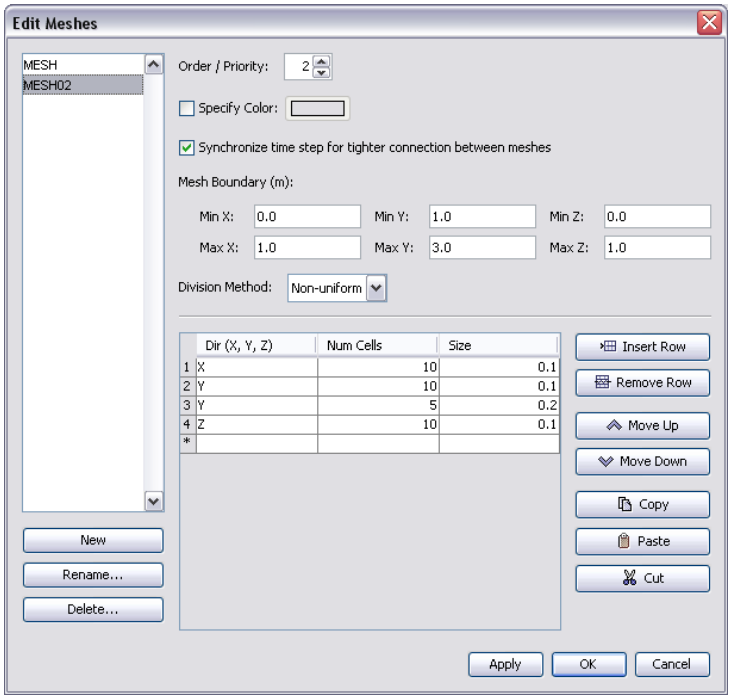


Figure 4.2. Defining properties of the nonuniform mesh

You can click (or type **Ctrl + R**) to reset the model. The resulting meshes are displayed below.

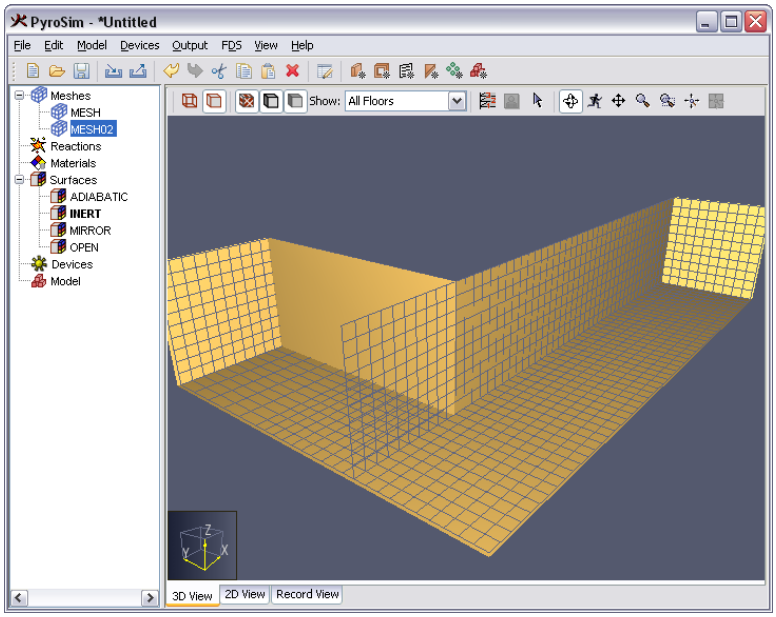


Figure 4.3. 3D display of first and second mesh

Using Multiple Meshes

The term “multiple meshes” means that the computational domain consists of more than one rectangular mesh, usually connected, although this is not required. In each mesh, the governing

equations can be solved with a time step based on the flow speed within that particular mesh. Some reasons for using multiple meshes include:

- Multiple meshes are required for parallel processing of FDS using the MPI option.
- If the geometry of the problem has corridors such as shown in Figure 4.3, using multiple meshes can significantly reduce the number of cells and the solution time.
- Because each mesh can have different time steps, this technique can save CPU time by requiring relatively coarse meshes to be updated only when necessary. Coarse meshes are best used in regions where temporal and spatial gradients of key quantities are small or unimportant.

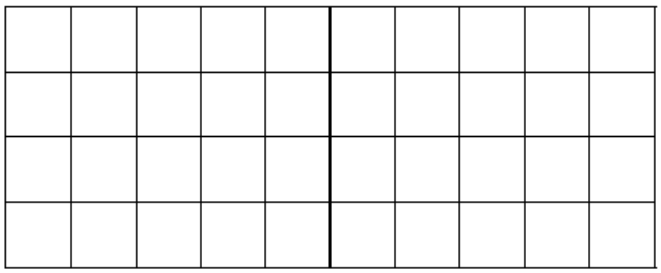
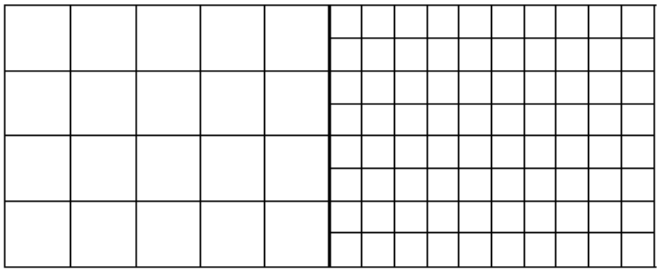
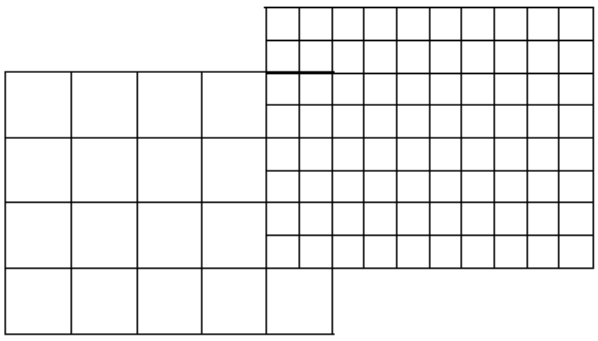
Meshes can overlap, abut, or not touch at all. In the last case, essentially two separate calculations are performed with no communication at all between them. Obstructions and vents are entered in terms of the overall coordinate system and need not apply to any one particular mesh. Each mesh checks the coordinates of all the geometric entities and decides whether or not they are to be included.

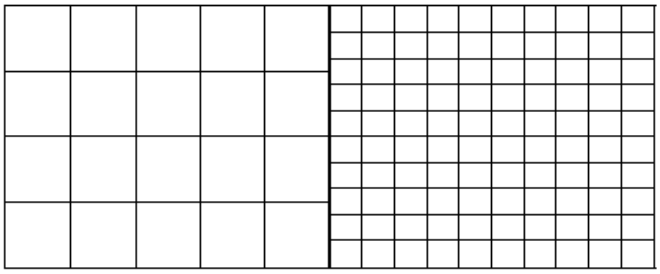
As described in the FDS 5 User Guide (McGrattan, et al., 2013), the following rules of thumb should also be followed when setting up a multiple mesh calculation:

- **Mesh Alignment**
  The most important rule of mesh alignment is that abutting cells ought to have the same cross sectional area, or integral ratios, as shown in Figure 4.4.
- **Mesh Priority**
  In general, the meshes should be entered from finest to coarsest. FDS assumes that a mesh with higher priority has precedence over a mesh with a lower priority if the two meshes abut or overlap.
- **Mesh Boundaries**
  Avoid putting mesh boundaries where critical action is expected, especially fire. Sometimes fire spread from mesh to mesh cannot be avoided, but if at all possible try to keep mesh interfaces relatively free of complicating phenomena since the exchange of information across mesh boundaries is not as accurate as cell to cell exchanges within one mesh.
- **Data Exchange**
  Information from other meshes is received only at the exterior boundary of a given mesh. This means that a mesh that is completely embedded within another receives information at its exterior boundary, but the larger mesh receives no information from the mesh embedded within. Essentially, the larger, usually coarser, mesh is doing its own simulation of the scenario and is not affected by the smaller, usually finer, mesh embedded within it. Details within the fine mesh, especially related to fire growth and spread, may not be picked up by the coarse mesh. In such cases, it is preferable to isolate the detailed fire behavior within one mesh, and position coarser meshes at the exterior boundary of the fine mesh. Then the fine and coarse meshes mutually exchange information.
- **Boundary Obstructions**
  If a planar obstruction is close to where two meshes abut, make sure that each mesh “sees” the

obstruction. If the obstruction is even a millimeter outside of one of the meshes, that mesh may not account for it, in which case information is not transferred properly between meshes.

- **Parallel Calculation**
  In a parallel calculation, it is recommended that the time steps in all meshes to be the same. This is the default setting in PyroSim and FDS 5 and provides a tighter connection between meshes. This option is selected by the Synchronize time step for tighter connection between meshes checkbox on the Edit Meshes dialog.
- **Trial and Error**
  Experiment with different mesh configurations using relatively coarse mesh cells to ensure that information is being transferred properly from mesh to mesh. There are two issues of concern. First, does it appear that the flow is being badly affected by the mesh boundary? If so, try to move the mesh boundaries away from areas of activity. Second, is there too much of a jump in cell size from one mesh to another? If so, consider whether the loss of information moving from a fine to a coarse mesh is tolerable.

	This is the ideal mesh alignment.
	This is allowed so long as there are an integral number of fine cells abutting each coarse cell.
	This is allowed, but of questionable value. PyroSim will warn if meshes overlap.



This is no longer allowed in FDS 5.1 and higher. PyroSim will warn against this mesh alignment.

Figure 4.4. Correct and incorrect mesh alignment

Additional Mesh Actions

To simplify working with multiple meshes, PyroSim provides the following additional mesh operations:

- **Split Mesh** splits selected meshes at a coordinate along a single axis.
- **Refine Mesh** makes selected meshes finer or coarser by a factor (e.g. refine a mesh by 4 to turn 1.0 meter cells into 0.25 meter cells).
- **Merge Meshes** combines two or more meshes into a single mesh.

To use any of the above actions, select one or more meshes, right-click to open a popup menu, then click the desired mesh action.

Chapter 5. Materials

To simulate a surface made of heat-conducting solids or a fuel you must specify a material that describes certain thermal properties and pyrolysis behavior. PyroSim offers two categories of materials: solid materials and liquid fuels.

To create a new material, you can use the **Edit Materials** dialog. On the **Model** menu, click **Edit Materials...**.

Solid Materials

Examples of solid materials include brick, gypsum board, and upholstery. To create a solid material:

1. In the **Edit Materials...** dialog, click **New...**
2. In the **Material Name** box, type the name of the new material
3. In the **Material Type** box, select `Solid`
4. Click **OK**

After following these steps, a default solid material will be created. Text entered in the **Description** box will not affect the simulation, but will be preserved in the FDS input file using the `FYI` field of the material. Including a description of the material is recommended.

The Thermal Properties tab provides the following options:

Parameter	Description
Density	The material's density.
Specific Heat	The material's specific heat. Specific heat can be specified as a function of temperature.
Conductivity	The material's heat conductivity. Conductivity can be specified as a function of temperature.
Emissivity	The material's emissivity. A value of 1.0 corresponds to maximum radiation.
Absorption Coefficient	This coefficient refers to the depth over which thermal radiation can be absorbed.

The **Pyrolysis** tab provides options to set the heat of combustion and add reactions that will be used to govern how the material burns. Each material can have a maximum of 10 reactions. To add a reaction, **click Add...**. This will open a dialog to edit the new reaction. It provides the following options:

On the **Rate** tab:

Parameter	Description
Reference Temperature	The temperature at which the material mass fraction decreases at its highest rate.
Heating Rate	The reaction rate at a given temperature.

Parameter	Description
Pyrolysis Range	The temperature range of
A (Pre-exponential Factor)	(parameter shown in equation)
E (Activation Energy)	(parameter shown in equation)
Mass Fraction Exponent	(parameter shown in equation)
Exponent	(parameter shown in equation)
Value	(parameter shown in equation)

On the **Byproducts** tab:

Parameter	Description
Heat of Reaction	Heat yield of this reaction. This must be a positive number.
Endothermic/Exothermic	Specifies if the heat yield is endothermic or exothermic.
Residue	The material that will be used to represent the residue. If there is only one material defined in PyroSim, this option will not be available.

Liquid Fuels

Examples of liquid fuels include kerosene and ethanol. To create a liquid fuel:

- In the **Edit Materials...** dialog, click **New...**
- In the **Material Name** box, type the name of the new material
- In the **Material Type** box, select `Liquid Fuel`
- Click **OK**

After following these steps, a default solid material will be created. Text entered in the **Description** box will not affect the simulation, but will be preserved in the FDS input file using the `FYI` field of the material. Including a description of the material is recommended.

The thermal properties tab for liquid fuels is identical to the thermal properties tab for solid fuels (see Solid Materials).

The **Pyrolysis** tab provides the following parameters:

Parameter	Description
Heat of Vaporization	Heat yield when this liquid fuel is converted to gas. This must be a positive number.
Endothermic/Exothermic	Specifies if the heat yield is endothermic or exothermic.
Heat of Combustion	The heat released when the liquid fuel combusts.
Boiling Temperature	The temperature at which the liquid fuel changes to a gaseous fuel.

Parameter	Description
Residue	The material that will be used to represent the residue. If there is only one material defined in PyroSim, this option will not be available.

Chapter 6. Surfaces

Surfaces are used to define the properties of solid objects and vents in your FDS model. The surface can use previously defined materials in mixtures or layers. By default, all solid objects and vents are inert, with a temperature that is fixed at the ambient temperature (set in the **Simulation Parameters** dialog. In addition to defining heat conduction in a solid, surfaces can also be used to define a burner, specify the ignition temperature for an object, give a vent a supply velocity, and set the many other properties supported by FDS.

To create, modify, and delete surfaces, you can use the **Edit Surfaces** dialog. To open the surface manager dialog, on the **Model** menu, click **Edit Surface Properties...**. The dialog in Figure 6.1 shows the dialog being used to edit an upholstery surface.

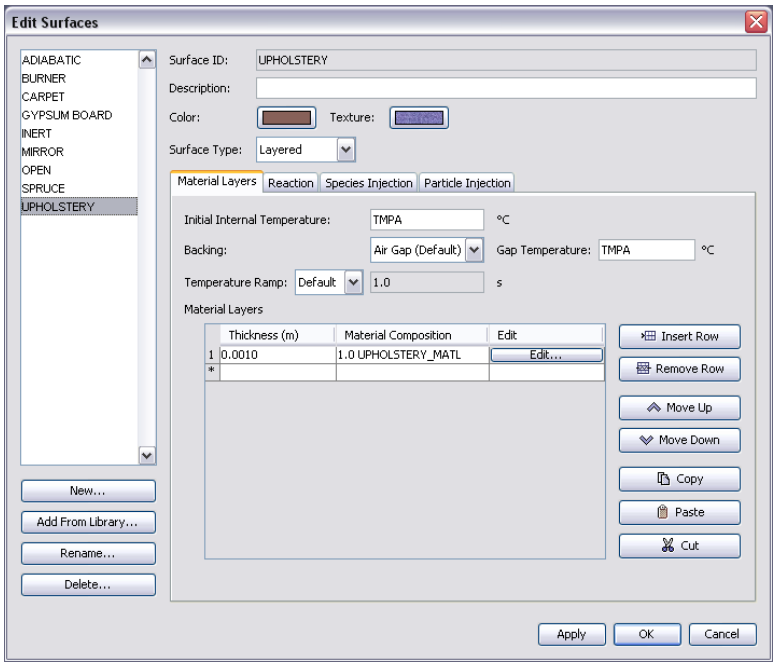


Figure 6.1. The Edit Surfaces dialog

Reserved Surfaces

There are four fundamental or “reserved” surface types: ADIABATIC, INERT, MIRROR, and OPEN. These surfaces cannot be changed and are present in every analysis.

- **ADIABATIC**
  This surface remains fixed at the ambient temperature. There is no heat transfer (radiative or convective) from the gas to an adiabatic solid.
- **INERT**
  This surface remains fixed at the ambient temperature. Heat transfer does occur from gases to INERT surfaces. This is the default surface in PyroSim.

- **MIRROR**[2]
  This surface is used only for vents on the exterior mesh boundary. A MIRROR is a no-flux, free-slip boundary that reverses flow. It is intended to be applied to an entire mesh boundary to symmetrically double the size of the domain.
- **OPEN**[2]
  This surface is used only for vents on the exterior mesh boundary. OPEN denotes a passive opening to the outside and is often used to model open doors and windows.
- **HVAC**
  This surface is used only for vents that are part of an HVAC system.

Surface Types

PyroSim aids the user by organizing the surface options into logical types, such as a `burner` to define a simple fire or a `layered` surface to represent a solid, heat conducting wall. The available surface types are described below.

Adiabatic

This surface type is identical to the built-in ADIABATIC surface type. It allows you to customize the description, color, and texture of the adiabatic surface described in Reserved Surfaces.

Inert

This surface type is identical to the built-in INERT surface type. It allows you to customize the description, color, and texture of the inert surface described in Reserved Surfaces.

Burner

This surface type represents a fire with a known heat release rate[3] or mass (fuel) loss rate.

Parameters for burner fires are arranged in two groups: heat release and particle injection. Heat release options:

Parameter	Description
Heat Release	
Heat Release Rate (HRR)	The heat release rate per unit area of this burner.
Mass Loss Rate	The mass loss rate per unit area of this burner.
Ramp-Up Time	At the beginning of the simulation, this surface will not be burning. This field allows you to describe how the heat release ramps up from ambient to the specified value.
Extinguishing Coefficient	This parameter governs the suppression of the fire by water. For more information, see section 10.7 of the FDS users guide.

[2] Vents of this type should not be toggled (deactivated or activated) during the simulation.
[3] The heat release rate for a burner surface is specified per unit area. A surface with a 500 kW/m^2 heat release rate applied to a 2.0 m^2 vent would result in a 1000 kW fire.

Parameter	Description
Temperature	
Surface Temperature	The surface temperature of this burner. The value TMPA represents ambient temperature.
Convective Heat Flux	The heat flux per unit area at the surface.
Ramp-Up Time	This field allows you to describe how the temperature ramps up from ambient to the specified value.
Other	
Emissivity	This parameter controls how the surface radiates heat. Using a value of 1.0 makes this surface a black body. Lower values increase the amount of radiated heat.

Particle injection options:

Parameter	Description
Emit Particles	Enable this option to emit particles from the surface.
Particle Type	Select a particle to emit. To create a new particle, click the **Edit Particles...** button.
Number of Particles per Cell	Controls the number of particles inserted per time step.
Insertion Interval	The frequency at which particles are inserted at the solid cells.
Mass Flux	For particles that have mass, this option provides an alternate way to control the number of particles inserted per second.

Heater/Cooler

This surface type represents a radiative heat source. The options are identical to the options for a burner without the heat release options. If the surface temperature is less than the ambient temperature, the surface will remove heat from the surrounding gases.

Supply

This surface represents a vent that injects air into the simulation domain. The parameters for supply surfaces are arranged in 4 groups: air flow, temperature, species injection, and particle injection.

Air flow options:

Parameter	Description
Specify Velocity	Use a constant velocity to define air movement through the vent.
Specify Volume Flux	Use a constant volume flux to define air movement through the vent.

Parameter	Description
Specify Mass Flux	Use a constant mass flux to define air movement through the vent.
Specify ... Individual Species	Define air movement through the vent using a table of extra species and their mass fluxes. This method requires a model that includes extra (non-reactive) species. Flux data is specified on the **Species Injection** tab.
Tangential Velocity	The tangential velocity of the air flow. The first parameter is the velocity in the x or y direction and the second parameter is in the y or z direction, depending on the normal direction of the vent. An example of tangential velocity is shown in Figure 6.2.
Ramp-Up Time	At the beginning of the simulation, vents with this surface will not be blowing. This parameter controls the time it takes to ramp the air flow up to the specified amount.
Wind Profile	The default wind profile is constant (Top Hat), to model wind conditions outdoors there are two additional options: parabolic and atmospheric. Parabolic produces wind with a parabolic profile whose maximum is the specified velocity. Atmospheric produces a wind profile of the form *u=u0(z/z0)^p*.
Atmospheric Profile Exponent	The term *p* in the atmospheric profile equation. This option is only available when atmospheric profile is selected.
Atmospheric Profile Origin	The term *z0* in the atmospheric profile equation. This option is only available when atmospheric profile is selected.

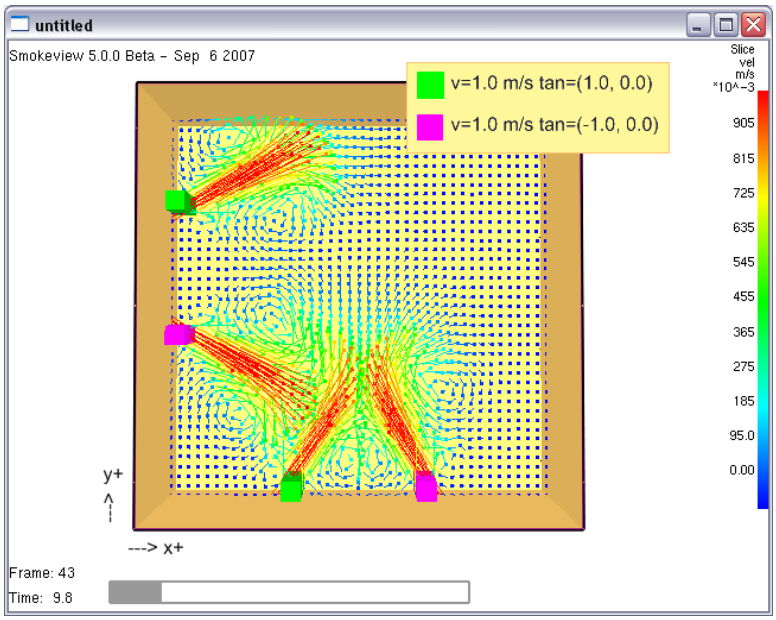


Figure 6.2. Effect of normal axis on the direction of tangential velocity

The temperature of the air injected by supply vents can be controlled using the following options:

Parameter	Description
Surface Temperature	The temperature of the injected air. The value TMPA represents ambient temperature.
Convective Heat Flux	The heat flux per unit area at the surface.
Ramp-Up Time	This field allows you to describe how the temperature ramps up from ambient to the specified value.
Other	
Emissivity	This parameter is not used for supply surfaces.

The species injection options are available if the **Specify Mass Flux of Individual Species** option in the **Air Flow** group is selected and there are extra, non-reactive species present in the simulation.

Particle injection options:

Parameter	Description
Emit Particles	Enable this option to emit particles from the surface.
Particle Type	Select a particle to emit. To create a new particle, click the **Edit Particles...** button.
Number of Particles per Cell	Controls the number of particles inserted per time step.
Insertion Interval	The frequency at which particles are inserted at the solid cells.
Mass Flux	For particles that have mass, this option provides an alternate way to control the number of particles inserted per second.

Exhaust

Exhaust surfaces can be used to remove gas from the simulation domain. The specification of their air movement parameters is identical to that of a supply surface, but instead of the velocity or flux driving air into the domain, they are pulling air out.

Fan

A fan is a special type of surface that more accurately simulates a fan than a supply surface. Fans must be attached to a thin obstruction that separates two pressure zones. For more information about fans, please refer to section 8.3.2 of the FDS User's Guide.

Layered

Layered surfaces are composed of one or more *material* definitions. Materials include solid and liquid substances such as concrete, pine, and ethanol. For more information about materials and how they can be specified in PyroSim, please refer to Chapter 5. This type of surface is ideal for walls and other objects

that are composed of real-world materials. This surface type can also be used to inject extra (non-reactive) species into the simulation.

Layered surfaces have five groups of options: material layers, surface props, reaction, species injection, and particle injection. The **Material Layers** group contains the following options:

Parameter	Description
Thickness	The thickness of this material layer.
Material Composition	Within a layer (row), you can specify multiple materials based on mass fraction. For example, to specify a layer that is just brick, type 1.0 BRICK (assuming you have created a material called BRICK). To specify a layer of wet brick, you could enter 0.95 BRICK; 0.05 WATER. Each material is separated by a semi-colon.
Edit	Click to specify the materials in this layer using an alternate table UI.

The **Surface Props** tab contains the following options:

Parameter	Description
Geometry	The effective geometry for heat transfer calculations.
Tangential Boundary Condition	Sets the slip condition at the boundary layer as either Default, Free Slip, or No Slip.
Roughness	Sets the surface roughness. This option is only available if the Tangential Boundary Condition is set to Default Slip.
Enable Leakage	This option allows you to select two pressure zones for leakage across the surface.
Initial Internal Temperature	Starting temperature inside the solid.
Backing	The backing of a surface is the boundary condition behind the surface. The default value, Air Gap represents an air gap, Exposed will allow the surface to transfer heat into the space behind the wall, and Insulated prevents any heat loss from the back of the material.
Gap Temperature	The temperature of air in the air gap. This option is only available when the Air Gap backing type is selected.
Temperature Ramp	Specifies the surface temperature ramp from ambient, to the specified surface temperature.

The reaction used to model a given surface can either be taken from the material specifications, or given explicitly by the surface. Manually specifying the parameters will produce a surface similar to a burner. You can edit this behavior using the reaction options:

Parameter	Description
Governed by Material	This surface's reaction will be controlled by the materials

Parameter	Description
	that it is constructed from.
Governed Manually	Override the default reaction behavior for this surface and specify the following parameters.
Heat Release Rate	The heat release rate per unit area of this surface.
Mass Loss Rate	The mass loss rate per unit area of this surface.
Ramp-Up Time	This field allows you to describe how the heat release ramps up from ambient to the specified value.
Extinguishing Coefficient	This parameter governs the suppression of the fire by water. For more information, see section 10.7 of the FDS User's Guide.
Burn Immediately	Select this option to create a surface that is initially burning.
Ignite at	Select this option to create a surface that will begin burning at a specified temperature.
Heat of Vaporization	Heat yield when this fuel is converted to gas.
Allow ... burn away	Surfaces of this type can be removed from the simulation after expending all available fuel.

You can inject extra (non-reactive) species into the simulation using the species injection options. To use these options, you must first specify extra species using the **Edit Extra Species** dialog. You can edit the following extra species options:

Parameter	Description
Inject by Mass Fraction	Select this option to specify species injection using mass fractions.
Inject by Mass Flux	Select this option to specify species injection using mass flux.
Species	This value cannot be edited. It displays the name of one of the species selected in the **Edit Extra Species** dialog.
Mass Fraction	The mass fraction of an extra species to inject. This option is only available if **Inject by Mass Fraction** is selected.
Mass Flux	The mass flux of an extra species to inject. This option is only available if **Inject by Mass Flux** is selected.
Ramp-Up Type	This field allows you to describe the function used to ramp up the injection rate from zero to the specified value.
Ramp Value	The time it takes to achieve the specified injection rate.

The particle injection parameters for layered surfaces are identical to those for burners.

Air Leak

Air leak surfaces can be used to create a permeable barrier between two pressure zones. The porous option is available to allow air movement to occur across the obstructions using this surface.

Adding Textures to Surfaces

You can add textures to surfaces to increase the realism of your model. Some default textures are provided or you can import your own. The Room Fire example demonstrates using a wood texture for a pine floor and hanging a picture on a wall. Your textures will be automatically displayed in PyroSim; to display textures in Smokeview, select **Textures** on the **Show/Hide** menu.

To define a texture:

1. On the **Model** menu, click **Edit Surfaces...**.
2. Either create or edit the surface to which you want to add a texture.
3. Click on the **Texture** box.
4. Either select a pre-defined texture or click the **Import...** button and select your own image file.
5. The image you selected will be displayed. Under the image, click the **Details** tab. Set the **Width** and the **Height** values to correspond to size to be used in the PyroSim model.
6. Click **OK** to close the **Textures** dialog.

The textured surface can now be used in either obstructions or vents.

Chapter 7. Geometry (Basic Concepts)

PyroSim provides tools to help the user rapidly create and organize model geometry.

Geometry can either be created through dialogs or by using the drafting tools in the **2D** or **3D** views as discussed in Chapter 8. There are typically three types of geometry that can be created in PyroSim:

- **Obstructions** – solid barriers to flow
- **Holes** – negative regions that carve holes in obstructions
- **Vents** – patches that define a sub region on an obstruction's surface that can have a different surface than the rest of the obstruction

The user can also organize the model by creating floors and groups. In addition, the user can assign background images to floors to aid in drafting.

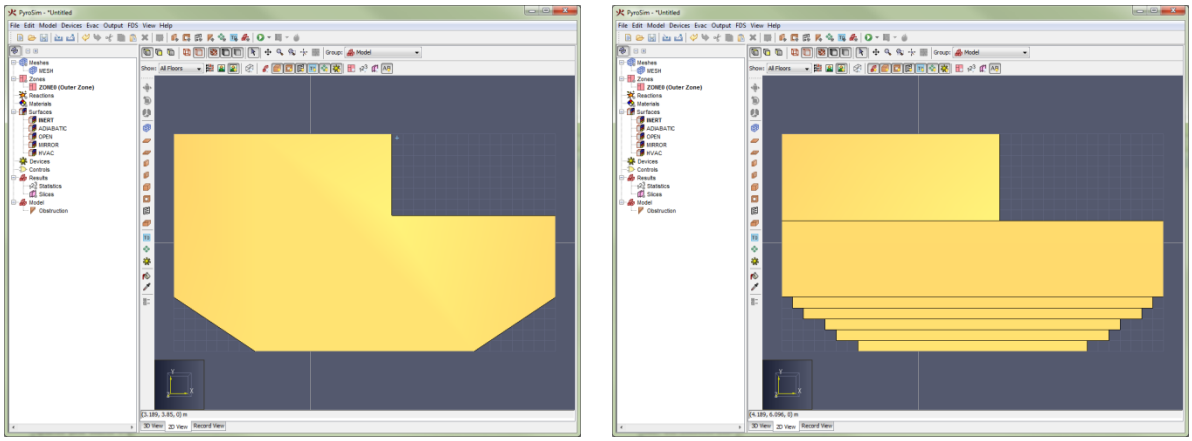
Obstructions

Obstructions are the fundamental geometric representation in FDS. In FDS, obstructions are rectangular, axis-aligned solids defined by two points. Surface properties are assigned to each face of the obstruction. In PyroSim, obstructions can take any shape, have any number of faces, and have different surfaces applied to each face. At the time of simulation, PyroSim will automatically convert the obstructions to axis-aligned blocks required by FDS as discussed in Angled Geometry on page 128.

FDS defines two types of obstructions:

- **Solid Obstructions** –obstructions that are at least one grid cell thick in all dimensions. FDS allows heat transfer calculations only on these types of obstructions. In addition, only these obstructions can have **vents** applied.
- **Thin Obstructions** –obstructions that have zero thickness in one dimension. These obstructions are mainly used to prevent flow. In addition, only this type of obstruction may be a **fan** as discussed in Chapter 6.

Figure 7.1 shows an example of a polygonal slab drawn in PyroSim and its conversion to blocks for use in FDS.



a b

Figure 7.1. Conversion of a slab obstruction to FDS blocks

Creating Obstructions

To create a new obstruction, either use an obstruction drawing tool as discussed in Chapter 8 or on the **Model** menu, click **New Obstruction...** or **New Slab…**.

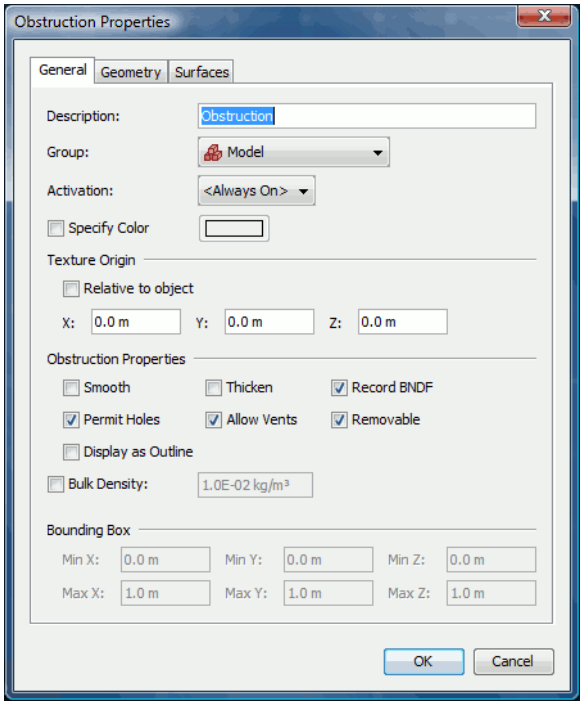


Figure 7.2. Obstruction dialog

General

This tab of the obstruction panel presents all options other than those controlling geometry and surface information. This includes activation events (conditions that can cause the obstruction to be added or removed from the simulation) and miscellaneous options such as color and smoothing.

Parameter	Description

Parameter	Description
Description	A human-readable description for the object. This value will not affect the result of the simulation.
Group	Controls the position of the object in PyroSim's tree view.
Activation	Bind this object to new or existing activation control logic. Activation control logic is used to add or remove the object based on time or measurement conditions. To learn more about activation events, please refer to Chapter 155.
Specify Color	Override the material colors for this object.
Texture Origin	
Relative to Object	When textures are attached to an object, they are tiled based on an origin point. By default, this point is the origin; **Relative to Object** makes the anchor point the minimum point of the object.
x,y,z	These values offset the texture origin based on the default texture origin. If the origin is relative to the object, leave at zero to use the object's min point.
Obstruction Properties	
Thicken	When this option is selected, this object will not be reduced to 2D faces by FDS. This is needed for obstructions to have vents attached.
Record BNDF	When this option is selected, this object is included in boundary data output.
Permit Holes	When this option is selected, holes can modify the geometry of this object.
Allow Vents	Makes it possible for this object to be the backing object for a vent.
Removable	Makes it possible for the object to be removed from the simulation by activation events or the BURN_AWAY surface option.
Display as Outline	Changes the appearance of this object.
Bulk Density	Use this option to override the amount of fuel provided by this object.

Geometry

This tab allows you to enter the min and max coordinates of the object. For more elaborate geometry, such as slabs, this tab may contain a table of points and extrusion options. Extrusion is the mechanism PyroSim uses to extend 2-dimensional objects along a vector - creating a 3-dimensional object.

Surfaces

By default, all six sides of an obstruction use the INERT surface. The Surfaces tab can be used to specify one surface to be used for all six sides of the object or assign surfaces on a per-face basis. Alternately,

surfaces can be “painted” using the **Paint Tool** as discussed in Painting Obstructions and Vents on page 76.

Holes

Holes are used to carve negative spaces out of obstructions. In FDS, holes are similar to obstructions in that they are defined as axis-aligned blocks. Like obstructions in PyroSim, however, holes can be any shape. PyroSim automatically converts them to blocks in the FDS input file.

PyroSim treats holes as first-class objects that be selected, deleted, and have other operations performed on them similar to obstructions as discussed in 0.

In the **3D** and **2D** views, holes appear as transparent objects. In addition, for display purposes only, PyroSim carves holes out of obstructions as shown in Figure 7.3. For complex holes or obstructions or large holes that span many obstructions, this process may be fairly slow. In these cases, hole-cutting display can be turned off by going to the **View** menu and deselecting **Cut Holes From Obstructions.**

While PyroSim will display obstructions with holes cut from them, it will NOT cut holes when creating the FDS input file. Instead, PyroSim converts holes and whole obstructions to blocks separately. PyroSim makes no attempt to remove obstructions that overlap holes from the FDS input file.

By default, all obstructions allow holes to be cut from them. To prevent an obstruction from allowing holes, edit the properties of the obstruction as discussed in 0 and deselect **Permit Holes**.

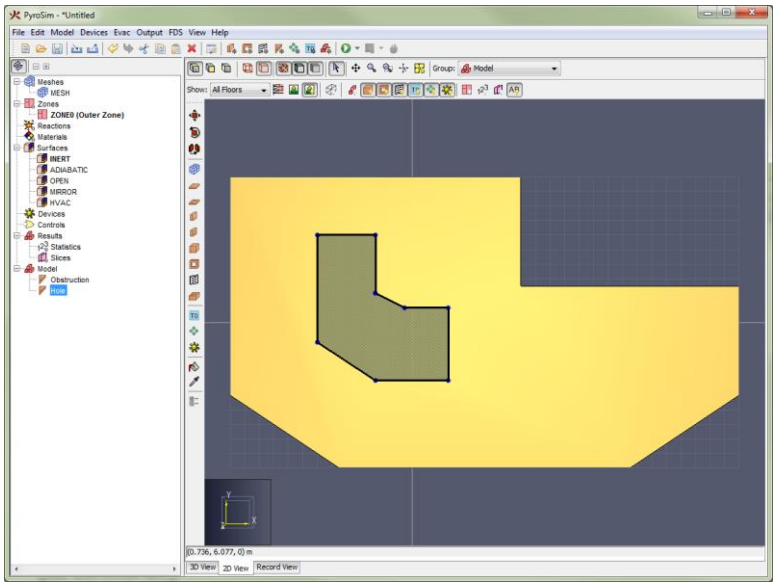


Figure 7.3. A slab obstruction with a hole cut from it

Creating Holes

Holes can either be drawn as discussed in Chapter 8 or can be created by opening the **Model** menu and clicking **New Hole…**. This will open the **Hole Properties** dialog as shown in Figure 7.4.

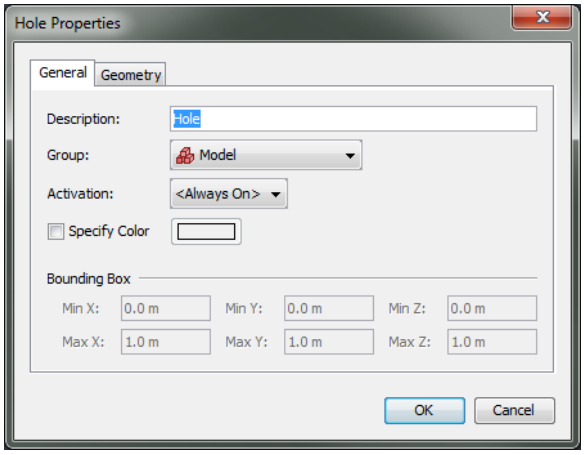


Figure 7.4. Hole Properties dialog

Like obstructions, holes can also be activated as discussed in Chapter 155. Holes can also have a color applied.

Vents

Vents have general usage in FDS to describe a 2D rectangular patch on the surface of a **solid** obstruction or on a mesh boundary as shown in Figure 7.5. A **vent** may have a different surface applied to it than the rest of the obstruction to which it is attached.

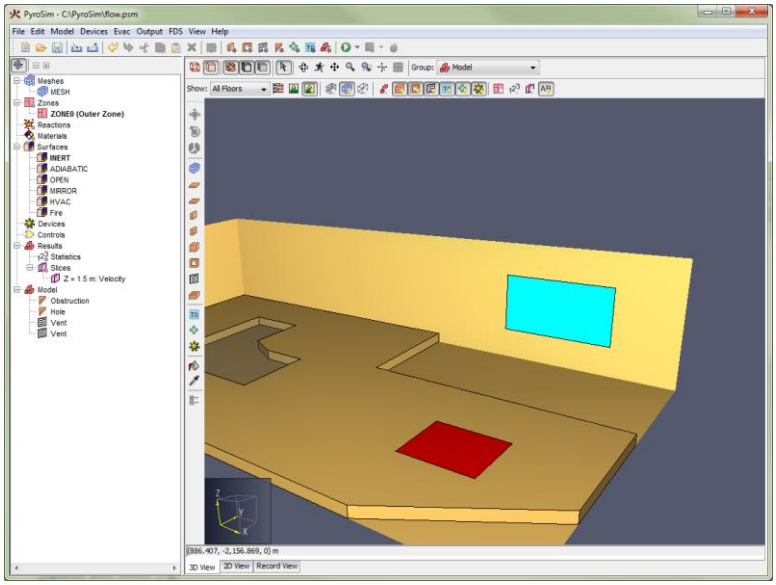


Figure 7.5. Vents on a mesh boundary and obstruction

Taken literally, a vent can be used to model components of the ventilation system in a building, like a diffuser or a return. In these cases, the vent coordinates form a plane on a solid surface forming the boundary of the duct. No holes need to be created through the solid; it is assumed that air is pushed out of or sucked into duct work within the wall.

You can also use vents as a means of applying a particular boundary condition to a rectangular patch on a solid surface. A fire, for example, is usually created by first generating a solid obstruction and then specifying a vent somewhere on one of the faces of the solid with the characteristics of the thermal and combustion properties of the fuel.

There are two reserved surface types that may be applied to a vent: `OPEN` and `MIRROR`. For more information on these types, see Chapter 6.

Creating Vents

Vents can either be drawn as discussed in Chapter 8 or be created by opening the **Model** menu and clicking **New Vent…**. This will open the **New Vent** dialog as shown in Figure 7.6. Like obstructions and holes, vents can also be activated, but only if the surface is not `MIRROR` or `OPEN`. With the exception of **Fire Spread**, the other properties are similar to obstructions. **Fire Spread** can be specified on vents using a burner surface (Chapter 6). This option simulates a radially spreading fire at the vent.

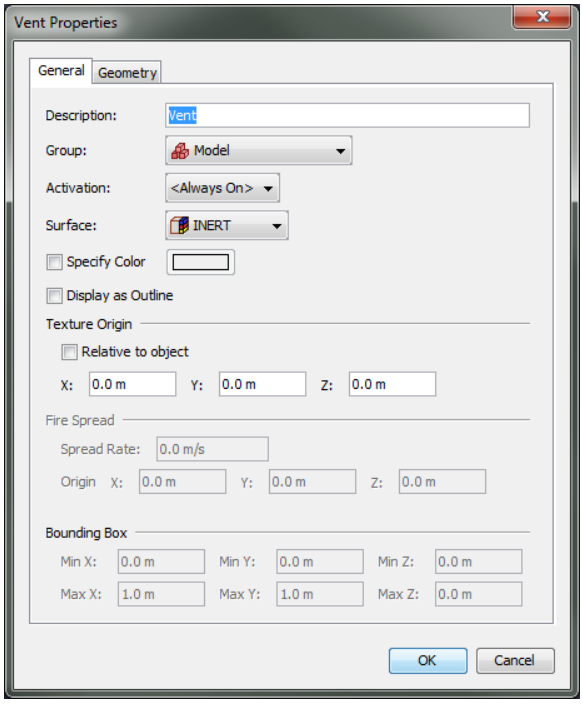


Figure 7.6. New Vent dialog

Groups

Groups can be used to hierarchically organize the model. Groups can only be seen in the **Navigation View.** The "Model" is the base group. Users can nest groups inside other groups, allowing the user to work with thousands of objects in an organized way. When the user performs an action on a group, that action will be propagated to all objects in the group.

Creating Groups

There are two ways to create a group:

- Right-click the desired parent group from the **Navigation View** and select **New Group…** from the context menu. This will create a child group in the selected parent.
- Click the **New Group…** button ( ) from the main tool bar.

Both of these actions will show the **Create Group** dialog as shown in Figure 7.7. This dialog allows the user to choose the parent group and name of the new group.

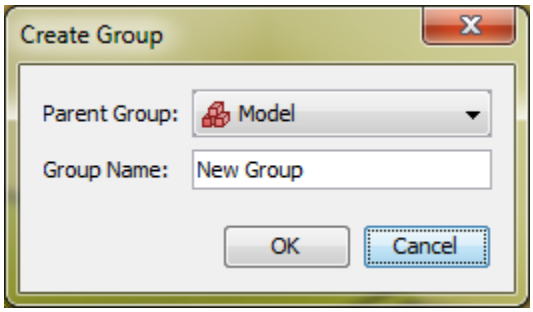


Figure 7.7. Create Group dialog

Adding Objects to Groups

There are several ways to add objects to a group:

- For **existing objects**, in the **Navigation View**, select the objects and drag them into the desired group as shown in Figure 7.8. Alternatively, right click them in any view and select **Change Group…**. In the **Change Group** dialog shown in Figure 7.9, select the desired group. If a new group is desired, select **New Subgroup** and specify a name. If this is chosen, a new group will be created under the specified existing group, and the selected objects will be moved to this new group.
- For **newly drawn objects**, in the **3D** or **2D** view select the desired group from the group drop-down above the view as shown in Figure 7.10. All newly drawn objects will be added to this group.

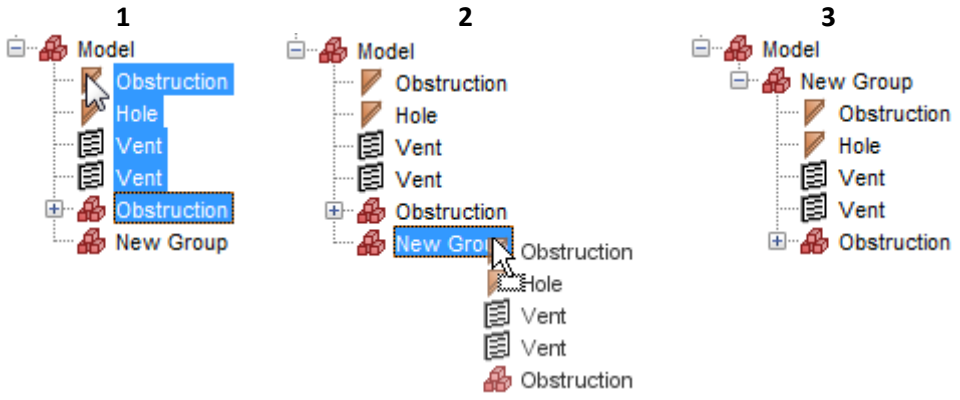


Figure 7.8. Dragging objects to a new group in the Navigation View

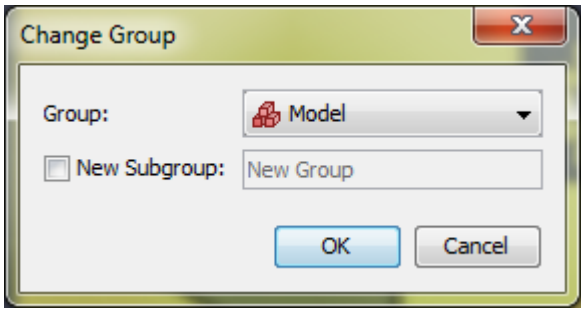


Figure 7.9. The Change Group dialog

Figure 7.10. The Group drop-down

Floors

Floors are used in PyroSim to quickly apply clipping filters to the scene to only show a portion of the model.

To define the floors in a model, go to the **2D** or **3D** View and click the **Define Floor Locations** button . This will display the **Manage Floors** dialog shown in Figure 7.11.

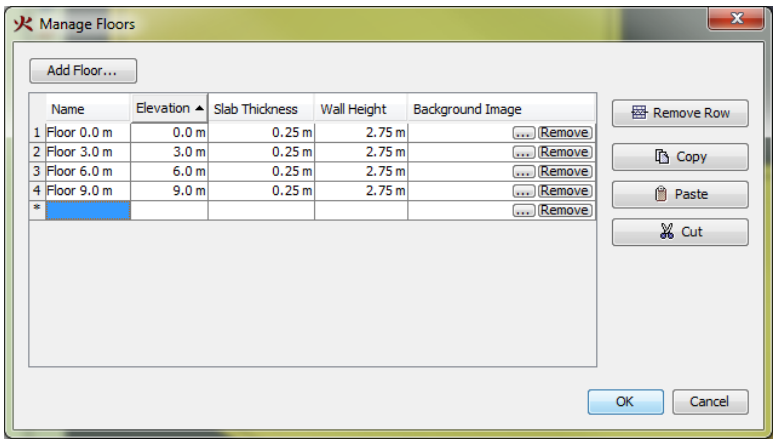


Figure 7.11. Manage Floors dialog

Floors are defined by the following properties:

- **Elevation** – the Z location at which walls and other obstructions will be drawn. This is the location where occupants would walk on that floor.
- **Slab Thickness** – the thickness of the slab for the floor. When the active floor is changed, this value is applied to the slab drawing tools but can be changed in the slab tool's properties. This

value is used such that the top of the slab is at the **elevation** and the bottom is at the **elevation** minus **slab thickness**.

- **Wall Height** – the height of the walls for floors. When the active floor is changed, this value is applied to the wall drawing tools but can be changed in the wall tool's properties. Walls are drawn from the **elevation** to the **elevation** plus **wall height**.
- **Background Image** – an image display along with the floor that can be traced over.

To add a new floor, click the **Add Floor…** button at the top of the **Manage Floors** dialog. This will show the **New Floor** dialog shown in Figure 7.12. By default this dialog will assume the user wants a floor above the previous floor using that floor's **slab thickness** and **wall height** properties. In this dialog, if the user enters a new **slab thickness**, the elevation will be automatically updated so the new floor does not overlap the others unless the user enters a specific value for the elevation. In addition, unless the user enters a specific name, a name will be automatically generated based on the elevation. Press **OK** to create the new floor.

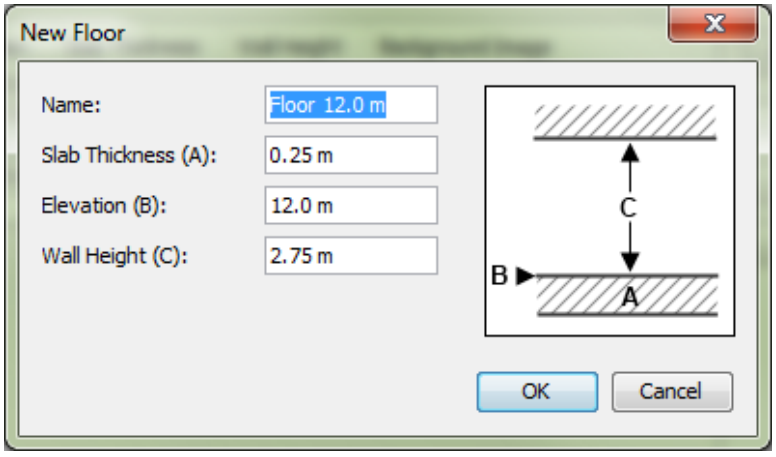


Figure 7.12. New Floor dialog

Press **OK** again in the **Manage Floors** dialog to commit the changes.

By default, the model contains one floor at **elevation** `0.0 m` with a **slab thickness** of `.25 m` and a wall height of `2.75 m`. Using these values leaves a distance of `3.0 m` from one floor **elevation** to another.

Once the floors have been defined, the user can filter the display to show either a single floor or all floors as shown in Figure 2.5. For most views, the Z clipping range for a particular floor is from the **floor elevation** minus **slab thickness** to **floor elevation** plus **wall height.** The Z clipping range works differently for the **top** camera of the **2D** view, however. In this view, the clipping is from the **elevation** of the floor BELOW to the **elevation** plus **wall height** of the current floor. This allows the geometry on the floor below to be snapped to in drawing geometry for the current floor. For this to be useful, however, the user may want to use wireframe rendering.

Adding a Background Image to a Floor

Each floor can have an associated background image. To add a background image to a floor, go to the **2D** or **3D** View, select a specific floor, then click the **Configure Background Image** button (alternately click the **Define Floor Locations** button, , and then in the Background Image column, select the Edit button). This will display the **Configure Background Image** dialog shown in Figure 7.13. You will be guided through the following steps:

1. Choose a background image file. Valid image formats are bmp, dxf, gif, jpg, png, tga, and tif.
2. Specify the **Anchor Point** for the image by clicking on the image. The **Anchor Point** is a point on the image at which the coordinates are specified in the model coordinate system. The model coordinates of the anchor point are not required to be at the origin.
3. Set the model scale. Select the **Choose Point A** button, then select the first point that will be used to define a length. Select the **Choose Point B** button and select the second point to define a length. Input the Distance between points A and B.
4. Use the sliding scale to change the image transparency.
5. If the image needs to be rotated, check the box next to **Dist. A to B** and enter an angle. This specifies the angle between the positive X axis and the vector from reference point A to B. For instance, if A->B should be aligned with the X axis, enter `0` for the angle or if A->B should be aligned with the positive Y axis, enter `90`.
6. Click **OK** to close the **Configure Background Image** dialog.

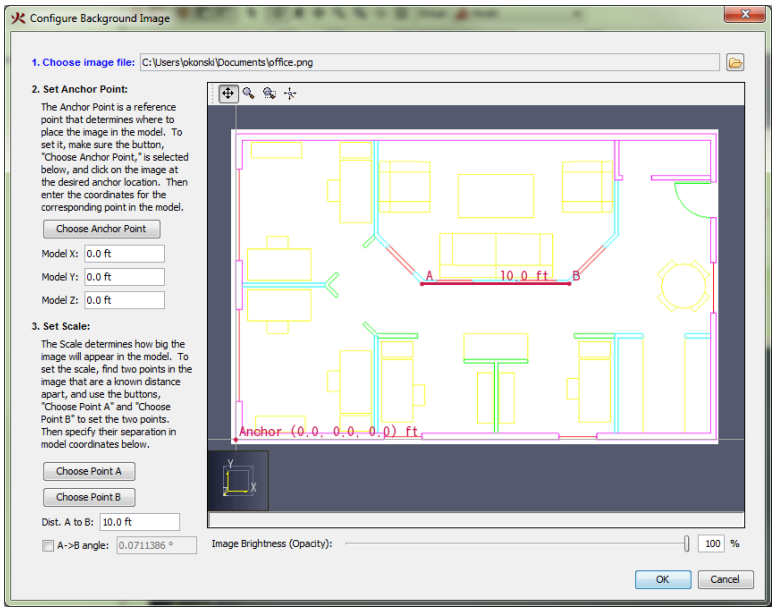

Configure Background Image
1. Choose image file:
C:\Users\pkonski\Documents\office.png
2. Set Anchor Point:
The Anchor Point is a reference point that determines where to place the image in the model. To set it, make sure the button, "Choose Anchor Point," is selected below, and click on the image at the desired anchor location. Then enter the coordinates for the corresponding point in the model.
Choose Anchor Point
Model X: 0.0 ft
Model Y: 0.0 ft
Model Z: 0.0 ft
3. Set Scale:
The Scale determines how big the image will appear in the model. To set the scale, find two points in the image that are a known distance apart, and use the buttons, "Choose Point A" and "Choose Point B" to set the two points. Then specify their separation in model coordinates below.
Choose Point A
Choose Point B
Dist. A to B: 10.0 ft
A->B angle: 0.0711386 °
A
10.0 ft
B
Anchor (0.0, 0.0, 0.0) ft
Image Brightness (Opacity):
100 %
OK
Cancel

Figure 7.13. Display of background image

Now, in the **3D** or **2D** views, when the user displays a specific floor, the background image for that floor will be displayed. To turn off the background images, go to the 2D or 3D View, and click the **Show Background Images** button next to the floors drop-down.

Chapter 8. Drawing in PyroSim

While not a full-fledged drafting application, PyroSim does provide useful drawing features, including the following:

- Snapping to grids, objects, and polar and orthographic constraints
- Precise keyboard entry
- 3D and 2D drawing, each having strengths over the other
- Mesh, Obstruction, Hole and Vent drawing tools
- Editing tools
- Transforming tools
- Surface painting tools for obstructions and vents

Drawing/Editing Tool Overview

PyroSim provides several drawing and editing tools. These tools are located on the **drawing toolbar** at the left side of the **3D** and **2D Views** as shown in Figure 8.1.

Figure 8.1. Drawing/Editing toolbar

Some of these tools allow a user to create and edit objects such as slabs and walls that are not constrained to the FDS mesh. In these cases, PyroSim will automatically convert the shapes to mesh-based blocks when the FDS input file is created. These blocks can be previewed by clicking the **Preview FDS Blocks** button ( ) on the filter toolbar above the **3D** or **2D View**. For information on block conversion, see Angled Geometry on page 128.

Selecting a Tool

To begin drawing or editing with a tool, the user can single-click the tool from the tool bar. Once the tool has finished drawing/editing its object, the last-used navigation tool is automatically selected.

If the user would like to create several objects with the same tool in succession, the desired tool can be **pinned** by clicking the tool's button twice. The button will show a green dot when pinned as shown in Figure 8.2.

Figure 8.2. Pinning a drawing/editing tool

Every time the same tool button is clicked, the pinned state of that tool will be toggled, so clicking the button again after pinning will disable pinning.

At any time, the current drawing/editing tool can be cancelled by pressing **ESC** on the keyboard. This will also cancel pinning and will revert back to the last-used navigation tool.

Tool Modes

Most drawing/editing tools require at least two points to be specified to complete its action, such as drawing the points for a wall or defining the extents of a box. These tools can operate in two modes:

- **Multi-click mode:** This mode allows the user to specify each desired point by single-clicking the left mouse button for each point. This mode also facilitates precise keyboard entry.
- **Click-drag mode:** This mode only allows the user to specify two points. When the left mouse button is pressed, the first point is committed. Then the mouse is dragged to the next location while still holding the left mouse button. The button is then released to specify the second point.

Tool Properties

Each tool has a set of properties that can be modified by clicking the **Tool Properties** button located at the bottom of the toolbar after selecting the desired tool. Options such as elevation, height, surface, and color can all be edited in the **Tool Properties** dialog.

Quick Actions

In addition to the tool properties, each tool also has additional quick actions. To show these actions, start the desired tool and then right-click in the **2D** or **3D View**. This opens a context menu with the quick actions. Figure 8.3 shows an example of the quick action menu for the wall tool.

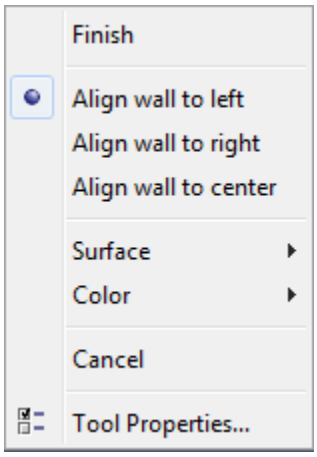


Figure 8.3. The quick action menu for the wall tool

This menu allows the user to perform actions specific to the tool, such as closing a polygon, picking a surface, setting wall alignment, accessing the tool properties, etc.

Snapping

Snapping is one way to precisely draw and edit objects. It is the process of finding some element in the scene, such as a **vertex** or **edge** close to the cursor, and snapping the cursor to that element like a magnet.

In PyroSim, snapping can be performed against the solution meshes, objects in the model, and orthographic constraints. The **2D View** additionally provides a sketch grid and polar (angle) constraints. If a snap point is found, an indicator dot will appear at the snap point as shown in Figure 8.4.

Figure 8.4. Snap indicator

By default, snapping is enabled. It can be disabled by holding **ALT** on the keyboard while using a drawing/editing tool.

Solution Mesh Snapping

If there are any solution meshes in the model (see Chapter 4), PyroSim can snap to them during drawing and editing. For each mesh that is visible, PyroSim can snap to its boundary edges, boundary faces, grid lines, and the intersections of the grid lines, depending on which mesh display filters are active as discussed in Filtering on page 11.

Sketch Grid Snapping (2D View Only)

PyroSim also provides a user-defined drawing grid, or sketch grid, in the 2D View as shown in Figure 8.5.

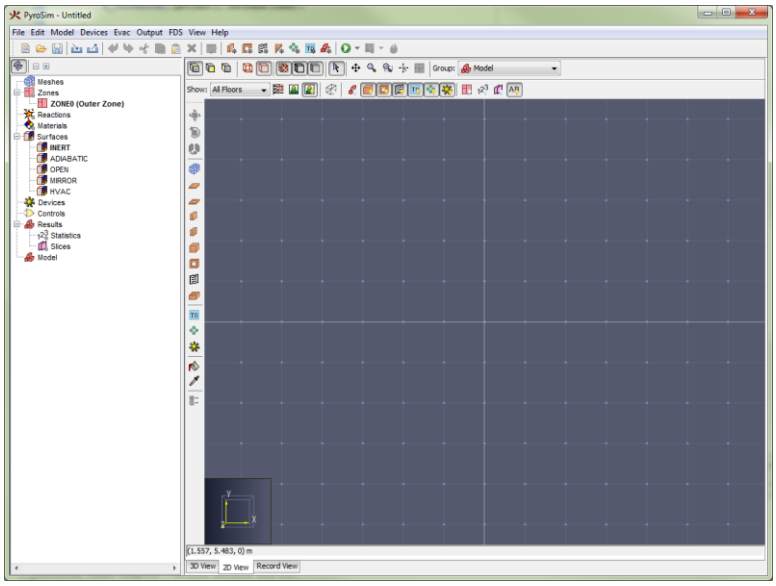


Figure 8.5. Sketch grid

When a new model is created, the sketch grid is visible and can be snapped to in the 2D view. The default spacing for the divisions is `1 m`, but can be changed by going to the **View** menu and clicking **Set Sketch Grid Spacing…**.

Once the user has created a solution mesh, PyroSim will automatically switch to solution mesh snapping and disable sketch grid snapping. In the **2D View**, PyroSim will only snap to the sketch grid or visible solution meshes. To switch which snapping is being used, on the **View** menu choose **Snap to Sketch Grid** or **Snap to Model Grids**. To disable grid snapping altogether, on the **View** menu choose **Disable Grid Snapping**.

Modeling Hint: In FDS, the spatial resolution of the solution is defined by the solution mesh, not the Sketch Grid. Using the solution mesh for **2D View** drawing ensures that the model geometry matches the FDS solution geometry. Some users create all model objects using mesh dimensions. While this leads to a "blocky" appearance, it does represent the true solution geometry and ensures there will be no unexpected gaps in the model.

Object Snapping

All objects displayed in the model can be snapped to when using the drawing/editing tools. There are three basic categories of geometry that can be snapped to on objects: **faces**, **edges**, and **vertices**. Objects can have any combination of types. If there are multiple types close to the cursor, PyroSim will give vertices precedence over edges and edges precedence over faces.

Constraint Snapping

Constraints are dynamic snapping lines that are only visible when the cursor is near them. They appear as infinite dotted lines as shown in Figure 8.6. PyroSim contains two types of constraints:

- **Orthographic:** These constraints allow the user to snap to a line parallel to the X, Y, or Z axis from the last relevant point. For instance, when drawing a polygonal slab, after each point specified, there will be three orthographic constraints extending from the last drawn point to aid in drawing the next point.
- **Polar (2D View Only):** These constraints are similar to orthographic constraints, but they are found at 15 degree increments from the current view's local X axis.

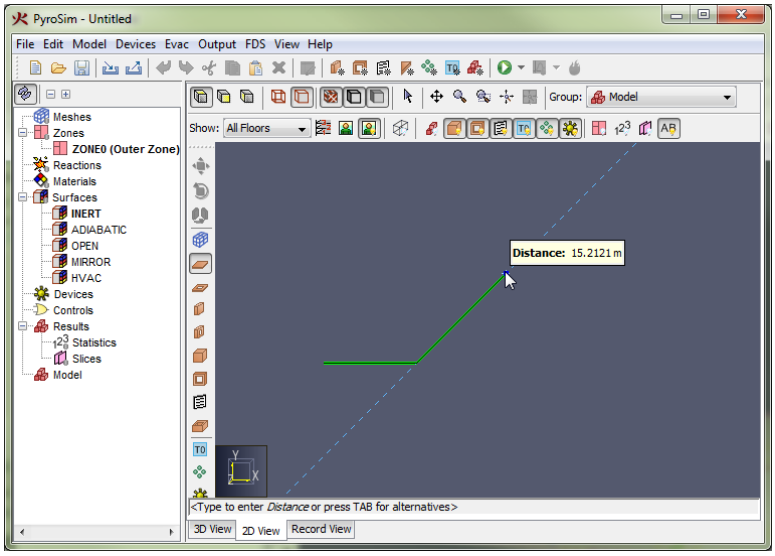


Figure 8.6. Polar constraint at an angle of 45 degrees

Constraint Locking

If a constraint is currently being snapped to, that constraint can be locked by holding **SHIFT** on the keyboard. While holding **SHIFT**, a second dotted line will extend from the cursor to the locked constraint (the first dotted line). This is useful for lining up objects along a constraint with other objects.

For instance, in Figure 8.7, a box already exists in the model. A second slab is being drawn such that the third point of the slab lines up with the right side of the first box. This was done as follows:

1) After the second point for the slab was clicked, the cursor was moved until the X-axis constraint became visible.
2) **SHIFT** was pressed and held on the keyboard to lock the constraint.
3) The cursor was moved to a point on the right edge of the box while still holding **SHIFT**.

NOTE: The distance displayed in the figure is the distance from the second point to the third point on the locked constraint and NOT from the second point to the cursor location.

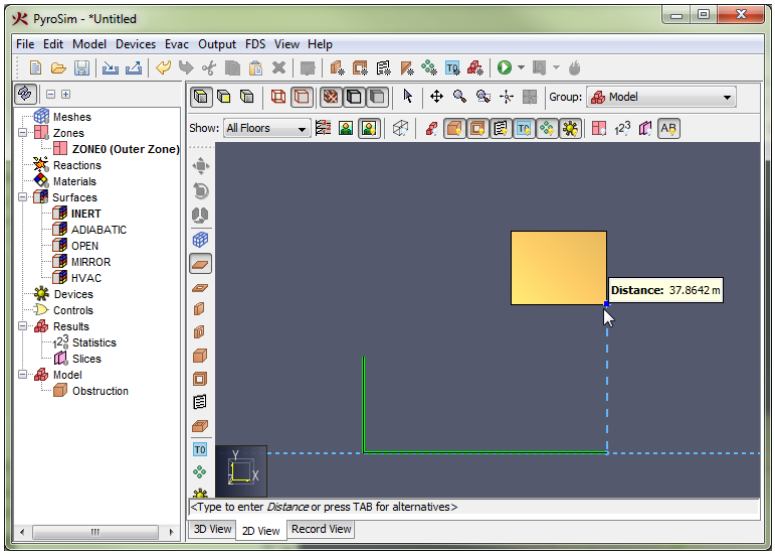


Figure 8.7. Locked constraint

Precise Keyboard Entry

While using a drawing/editing tool, a popup window may appear next to the cursor, such as in Figure 8.8. This window shows the value used to determine the next point or value for the current tool. In this figure the value is the **Distance** from the previous point along the vector from the previous point to the current cursor location. For other tools, this value may be angle or relative offset, etc.

The value is editable if the status bar at the bottom of the **3D** or **2D View** indicates it is. For instance, in the figure, the status bar says "<Type to enter Distance or press TAB for alternatives>". If the user starts typing, the popup window will be replaced with an editing window as shown in Figure 8.9. If the user presses **ENTER**, the typed value will be committed. If the user presses **ESC** instead, the keyboard entry will be cancelled.

Pressing **TAB** cycles through alternate input methods to determine the next value. For instance, pressing **TAB** with the wall tool allows the user to enter a relative offset from the last point instead of a distance. Pressing **TAB** a second time allows the user to enter an absolute position for the next point, and pressing **TAB** a third time will cycle back to the distance input.

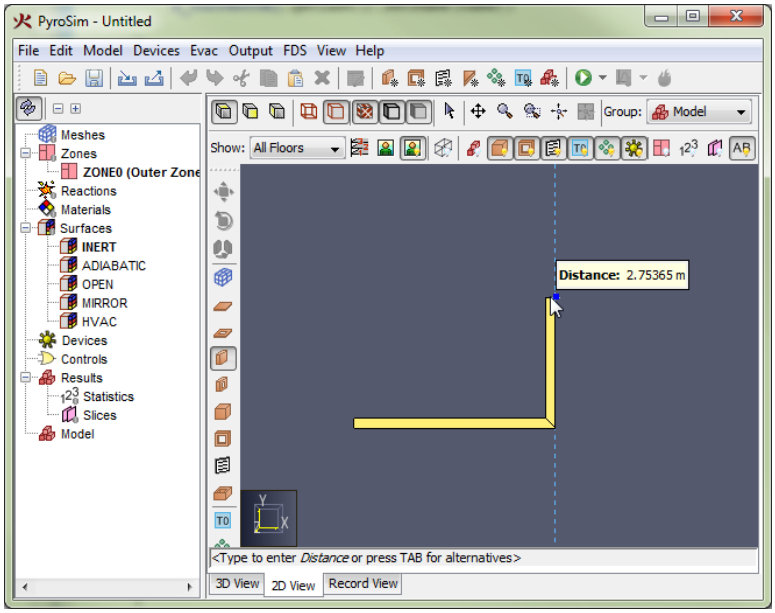


Figure 8.8. Tool editor window

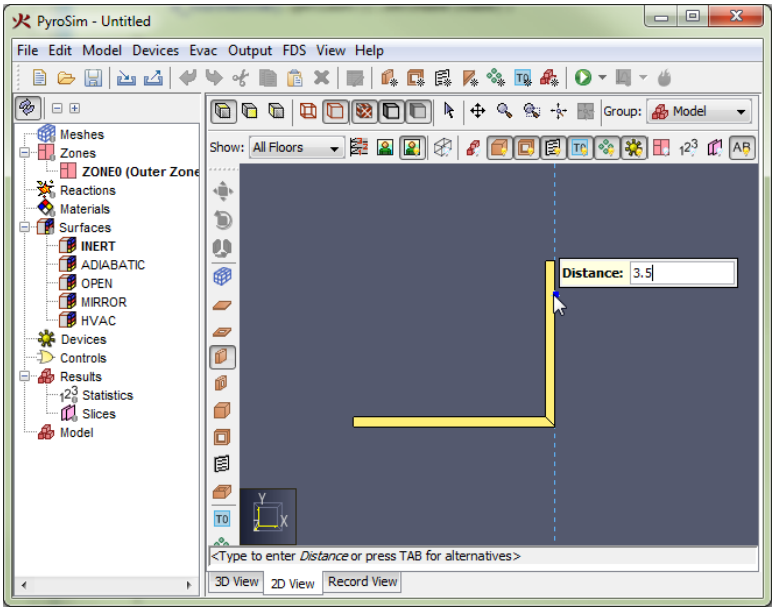


Figure 8.9. Precise keyboard entry

Precise keyboard entry may be easiest for some users when using the **multi-click mode** of drawing rather than using the **click-drag mode**. Using multi-click allows both hands to be used to type as opposed to click-drag, which requires one hand to remain on the mouse.

2D versus 3D Drawing

There are some key differences between drawing in the **2D** and **3D Views**. The **2D View** is useful when drawing should be restricted to one pre-defined plane. It is also useful for lining up objects along the X, Y, or Z axes. The **3D View** is useful when an object such as a vent or solid-phase device needs to be

snapped to the face of an obstruction or vent or if the user would like to build objects by stacking them on top of one another.

2D View Drawing

When drawing in the **2D View**, the drawing will always take place in the drawing plane specified in the tool properties, and snapping is only performed in the local X and Y dimensions. The local Z value will remain true to the drawing plane. In addition, if a tool has some sort of height or depth property, the tool will also remain true to that value. Figure 8.10 shows two slabs that have been drawn in the **top** view, one at Z = 0 m and the other at Z = 1.5 m. While snapping was used to partially align the objects, they both remain in the Z planes specified in their tool properties.

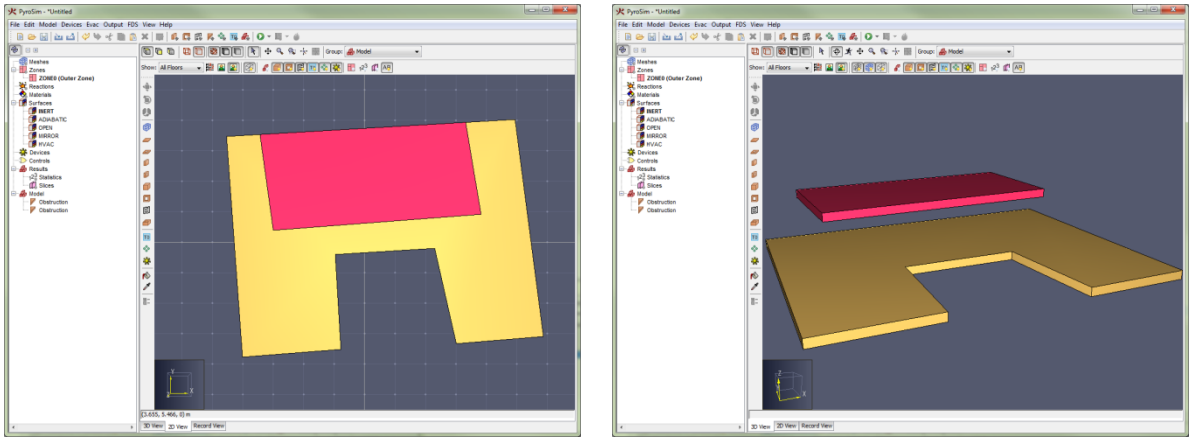


Figure 8.10. Slabs in different planes aligned in the 2D View

3D View Drawing

The **3D View** uses snapping in all three dimensions, causing tool properties to be interpreted more loosely. The drawing plane and depth properties for a drawn object are context-sensitive in the **3D View**. When using tools such as the slab tool, the first clicked point determines the drawing plane. If, on this first click, another object is snapped to, the drawing plane is set at the Z location of that snap point. The tool properties' drawing plane is only used if nothing is snapped to on the first click.

This 3D snapping feature of the **3D View** is useful for drawing vents on obstructions and attaching solid-phase devices to obstructions as shown in Figure 8.11.

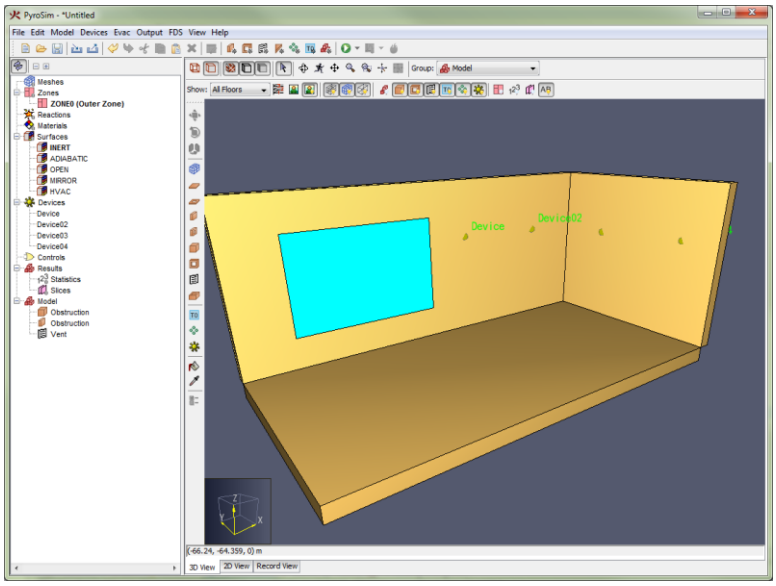


Figure 8.11. Vent and devices attached to a wall

The 3D snapping feature is also useful for stacking objects, as shown in Figure 8.15. In this figure, the drawing plane was never changed. All the objects were stacked on top of each other using snapping.

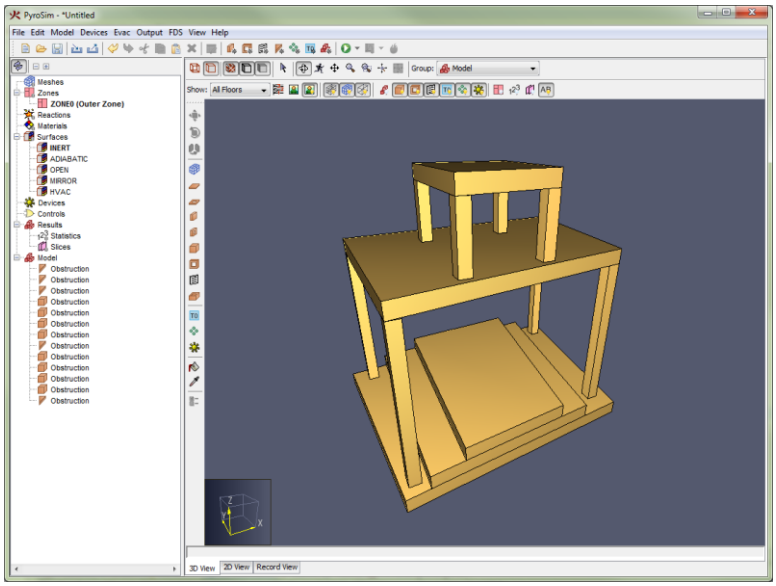
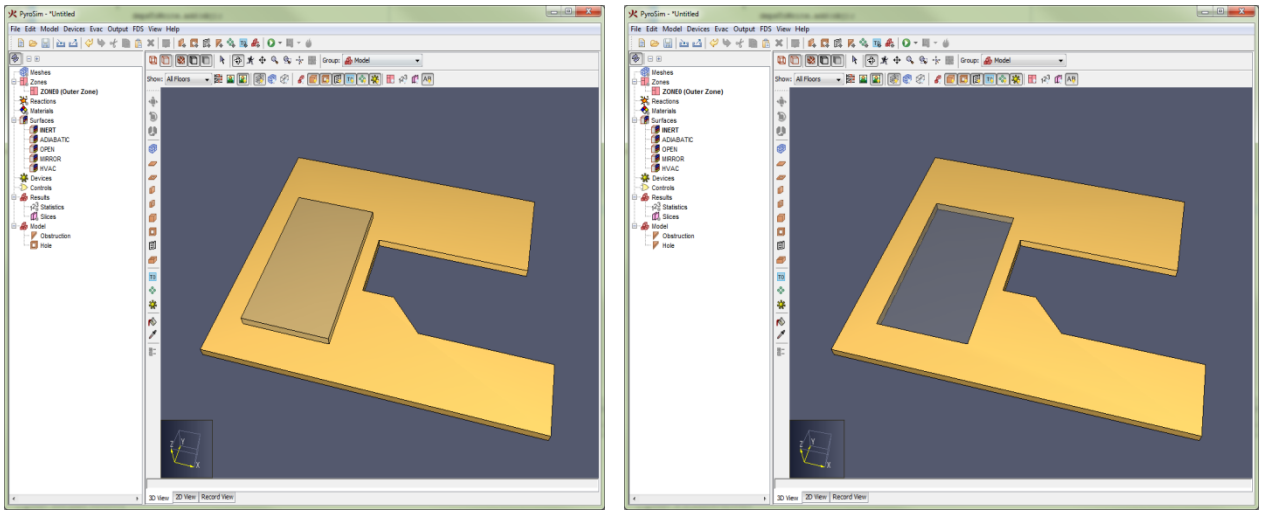


Figure 8.12. Objects stacked in the 3D View

Holes in the 3D View

While stacking can be useful for obstructions, a user must be more careful when drawing holes in the **3D View**. For instance, with the **slab hole tool** and **block hole tool**, the user will need to change the extrusion direction to properly direct the hole into the obstruction. For instance, if the user draws a slab obstruction in the **3D View** and then draws a slab hole while snapping to the obstruction, the hole will be stacked on top of the obstruction without cutting a hole as shown in Figure 8.13 (A). To draw this

properly, the user would need to change the extrusion direction when drawing the hole by pressing **CTRL** on the keyboard or changing it through the tool's right-click menu. This will result in a proper hole as shown in Figure 8.13 (B). This is not a problem in the **2D View** since it always uses the drawing plane set in the tool properties instead of stacking the objects.



A **B**

Figure 8.13. Improper vs proper hole drawing in the 3D view

Projected Drawing in the 3D View

Once the drawing plane for a tool has been established by the first click, the tool can still determine the next points by snapping to objects in another plane. In this case, the snapped points will be projected to the drawing plane for the current tool. A dotted line will show how the snapped point was projected to the plane. For instance, Figure 8.14 shows a new slab being drawn in the Z = 1.5 plane. A slab below the new slab is being snapped to determine the new slab's points.

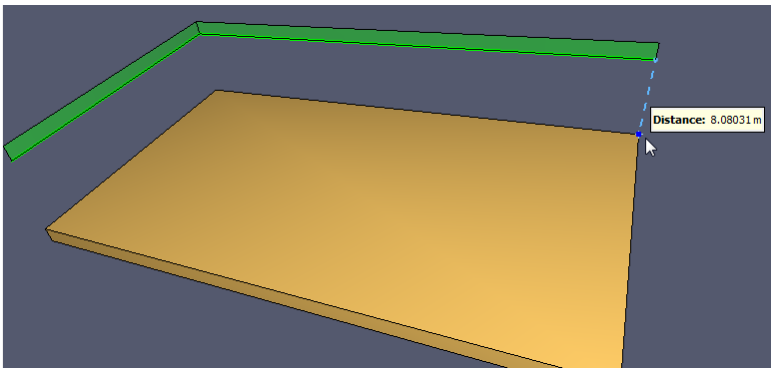


Figure 8.14. Snapping to another plane in the 3D View

Obstruction Drawing Tools

There are four tools that can draw obstructions (for more information on obstructions, see Obstructions on page 41).

- **Slab Obstruction Tool:** Used to draw the slab for a floor
- **Wall Obstruction Tool**: Used to draw a wall
- **Block Obstruction Tool:** Used to fill grid cells with obstructions
- **Room Tool:** Used to draw a rectangular room

For all the obstruction tools, the tool properties dialog ( ) will appear similar to that in Figure 8.15. The only section of the dialog that will change between these tools is the geometry, such as **Z Location** and **Thickness**. All other properties, including name, surface, color, and obstruction flags appear in all obstruction dialogs. These parameters control the properties that will be applied to the next drawn obstruction.

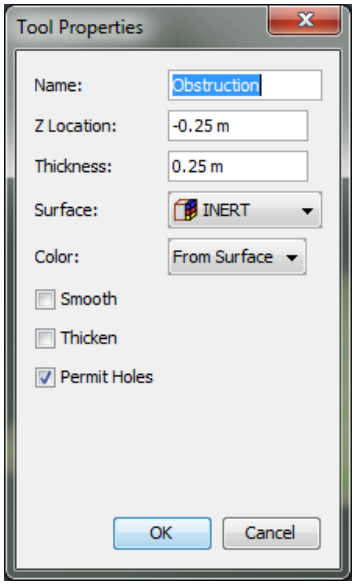


Figure 8.15. Tool properties dialog for obstructions

The surface and color of the next obstruction can also be set via the right-click menu for the tool.

Slab Obstruction Tool

A slab is an extruded polygonal object as shown in Figure 8.16 that can be used to draw the slab for a floor in a building.

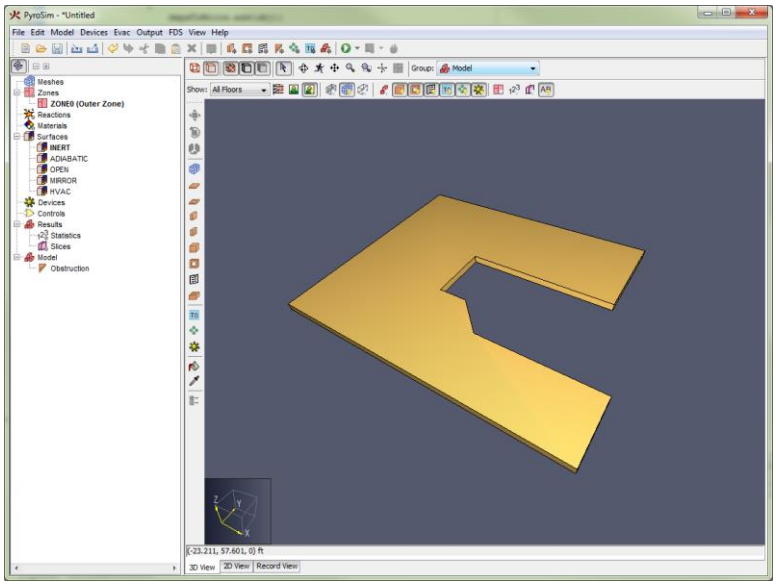


Figure 8.16. A polygonal slab obstruction

The slab obstruction tool adds two additional properties to the tool dialog for obstructions:

- **[X,Y, or Z] Location:** the drawing plane for the slab. When the active **floor** is changed, this is set to the floor's **elevation** minus the floor's **slab thickness**.
- **Thickness**: the thickness of the slab. When the active **floor** is changed, this is set to the floor's **slab thickness**. If this value is positive, the slab is extruded toward the camera. If it is negative the slab is extruded away from the camera. The extrusion direction can be toggled by pressing **CTRL** on the keyboard or from the tool's right-click menu.

To draw the polygon vertices of the slab obstruction, perform the following:

1. Select the **Slab Obstruction Tool** from the drawing toolbar.
2. Define the slab points using one of the two tool modes:
   - Use **Click-drag mode** to draw an axis-aligned box between two points.
   - Use **Multi-click mode** to click several points defining the polygonal boundary for the slab. The slab will automatically close if the first point is clicked again or if the user selects **Close** from the right-click menu.

Wall Obstruction Tool

The wall obstruction tool can be used to draw multi-segmented walls as shown in Figure 8.17. In this figure, there is only one wall. The user specifies a path along the floor from which the wall is extruded up. The wall can be aligned to the left, right, or center of the drawn path.

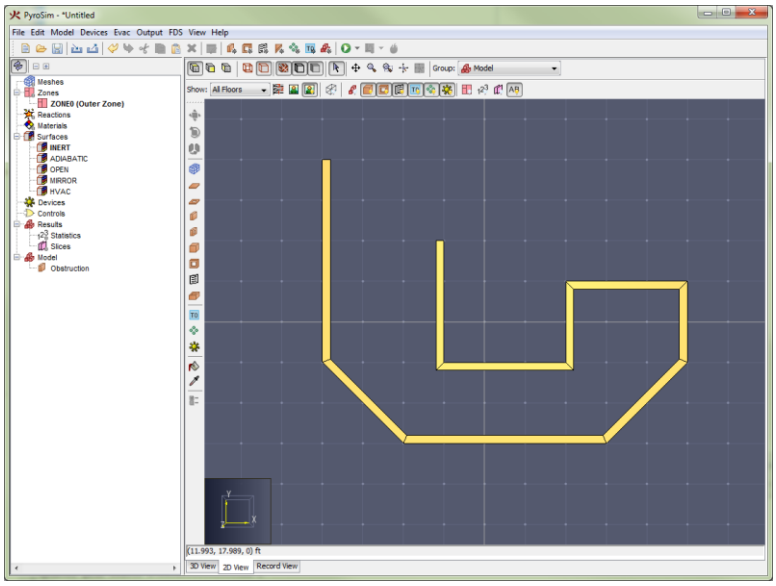
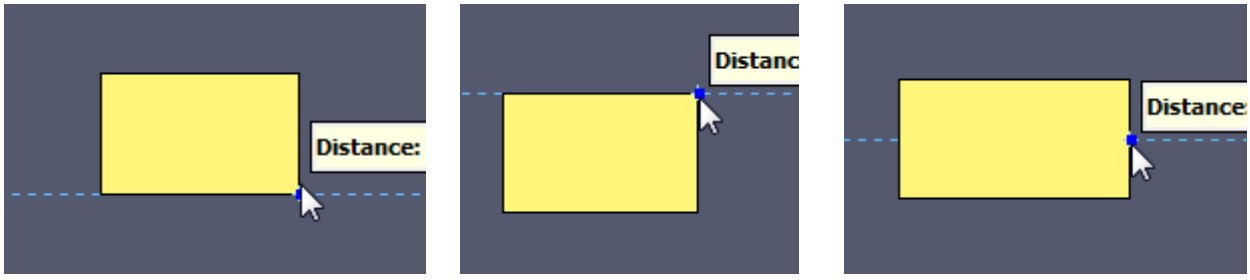


Figure 8.17. A wall obstruction

The wall obstruction tool adds three properties to the tool properties dialog:

- **Z Location:** the drawing plane for the bottom of the wall. When the active **floor** is changed, this is set to the floor's **elevation** property.
- **Height:** the height of the wall. This must be positive. When the active **floor** is changed, this is set to the floor's **wall height** property.
- **Wall Thickness:** the thickness of the wall.

The alignment of the wall can be controlled through the right-click menu for the tool or can be cycled by pressing the **CTRL** key on the keyboard. The alignment options are shown in Figure 8.18.



Left-aligned **Right-aligned** **Center-aligned**

Figure 8.18. Wall alignment options

To draw a wall, perform the following steps:

1. Select the **Wall Obstruction Tool** from the drawing toolbar.
2. Define the wall points using one of the two tool modes:
    - Use **Click-drag mode** to draw a single wall segment.

- Use **Multi-click mode** to click several points, with wall segments between each pair of points. If the first clicked point is clicked again after drawing at least two segments or **Close** is chosen from the right-click menu, the tool will draw one last segment from the last clicked point to the first point and finish. Alternately, the wall can be ended at the last clicked point by choosing **Finish** from the right-click menu.

Block Obstruction Tool

The **Block Obstruction Tool** can be used to quickly fill grid cells with blocks as shown in Figure 8.19 or place a block with a single click.

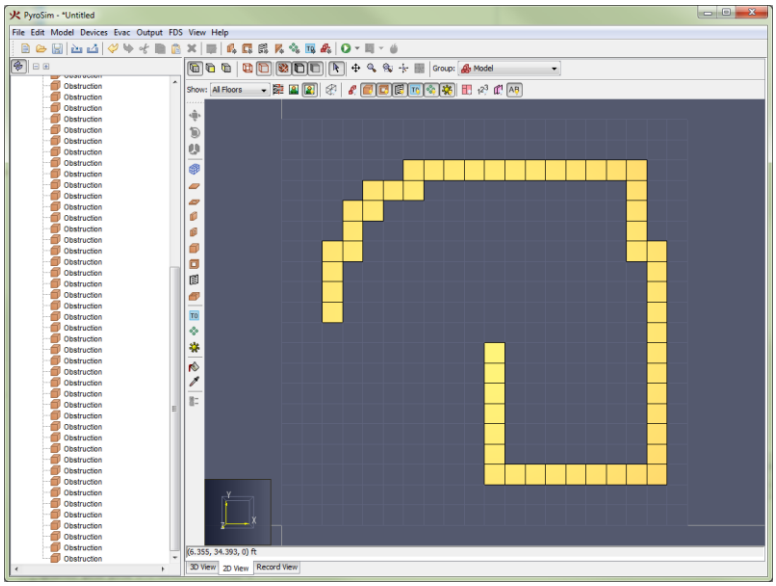


Figure 8.19. Blocks drawn with the block obstruction tool

This tool adds the following properties to the tool properties dialog:

- **[X,Y, or Z] Location:** the drawing plane. When the active **floor** is changed, this is set to the floor's **elevation**.
- **Height:** the distance the block is extruded toward or away from the camera. When the active **floor** is changed, this is set to the floor's **wall height** property.
- **Size:** the size of the block when there is no snap grid under the cursor.

In addition, the extrusion direction for the block can be toggled by pressing **CTRL** on the keyboard or through the right-click menu for the tool.

To create blocks using this tool, perform the following:

1. Select the **Block Obstruction Tool** from the drawing toolbar.
2. If a solution mesh or sketch grid is visible, either click the desired cell to fill or click-drag the mouse across the grid to "paint" blocks. The depth of the cells will not necessarily be the depth

of a cell in the filled mesh, however. The depth is strictly controlled by the **height** property for the tool.

Room Tool

The **room tool** can be used to draw a rectangular room using one closed wall as shown in Figure 8.20.

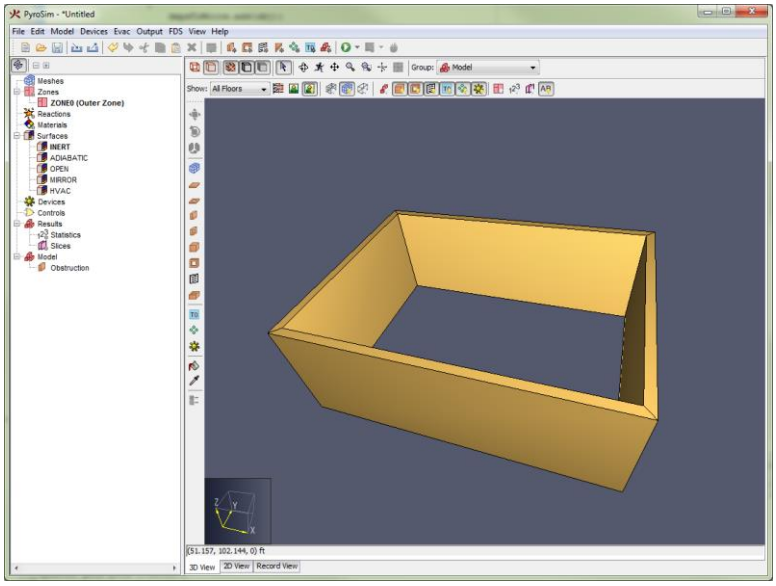


Figure 8.20. A room drawn with the room tool

The **room tool** contains the same properties as the **wall obstruction tool**.

To draw a room with the **room tool** perform the following:

1. Select the **Room Tool** from the drawing toolbar.
2. Use C**lick-drag mode** or **Multi-click mode** to draw two points defining the extents of the room.

Hole Drawing Tools

There are three tools that can draw holes (for more information on holes, see Holes on page 44).

- **Slab Hole Tool:** Used to draw a hole in a floor slab.
- **Wall Hole Tool**: Used to draw an opening in a wall, such as for a doorway or window.
- **Block Hole Tool:** Used to fill grid cells with holes.

All these tools work the same as their obstruction counterparts, but they do not have the properties specific to obstructions, such as the surface or obstruction flags.

Vent Tool

There is only one tool for drawing vents (for more information on vents, see Vents on page 45). PyroSim only allows vents in an X, Y, or Z plane. Vents cannot currently be drawn off-axis like walls can.

Vents also must be attached to solid obstructions (at least one grid cell thick). This is easily accomplished by drawing the vent in the **3D View** (see 2D versus 3D Drawing on page 57).

To draw a vent, perform the following:

1. Select the **Vent Tool** from the drawing toolbar.
2. Use **Click-drag Mode** or **Multi-click Mode** to specify two points that define the extents of the vent.
    - When this is performed in the **2D View**, the vent will be drawn in the plane specified in the tool properties and is view-dependent (in the **top** view, the vent will be drawn in the **XY** plane, in the **front** view, the vent will be drawn in the **XZ** plane, etc.).
    - If the vent is drawn in the **3D View**, the vent will be drawn in the plane containing the two specified points. If the two points are not in an axis-aligned plane, an axis-aligned plane will be chosen containing the first point while maximizing the area of the vent.

Solution Mesh Tool

Solution meshes can also be drawn in PyroSim with the **solution mesh tool** as shown in Figure 8.21.

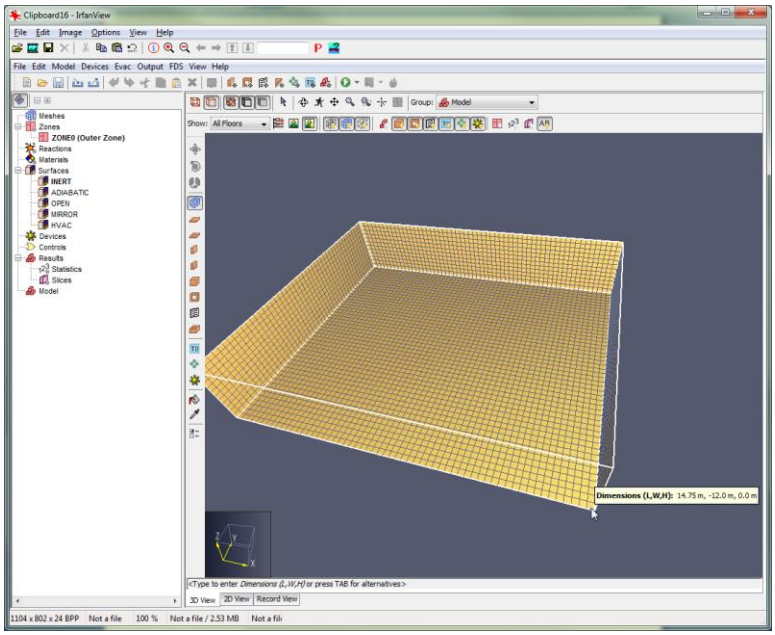


Figure 8.21. A solution mesh drawn with the mesh tool

The solution mesh tool has the following tool properties:

- **[X,Y, or Z] Location**: The drawing plane containing the bottom of the mesh. When the active **floor** is changed, this is set to the floor's **elevation** minus the floor's **slab thickness**.
- **Height:** The depth of the mesh in the current view. When the active **floor** is changed, this is set to the floor's **wall height** plus the floor's **slab thickness**.

- **Cells:** Define how cells are generated as the mesh is being drawn. This can be one of two values: **fixed size**, or **fixed count**:
  - **Fixed Size:** each cell will have a fixed size. While drawing the mesh, the mesh boundary will snap to integer multiples of the cell size. Figure 8.22 (A) shows a mesh drawn with fixed cell sizes of `.25 x .25 x .25 m`. As can be seen, the grid has been snapped away from the cursor the nearest multiple of `.25 m`.
  - **Fixed Count:** each dimension of the drawn mesh will be evenly divided to have the specified number of cells. Figure 8.22 (B) shows a mesh drawn with fixed counts of `10 x 10 x 10`. As can be seen, the cells are stretched to match the drawn mesh boundary.

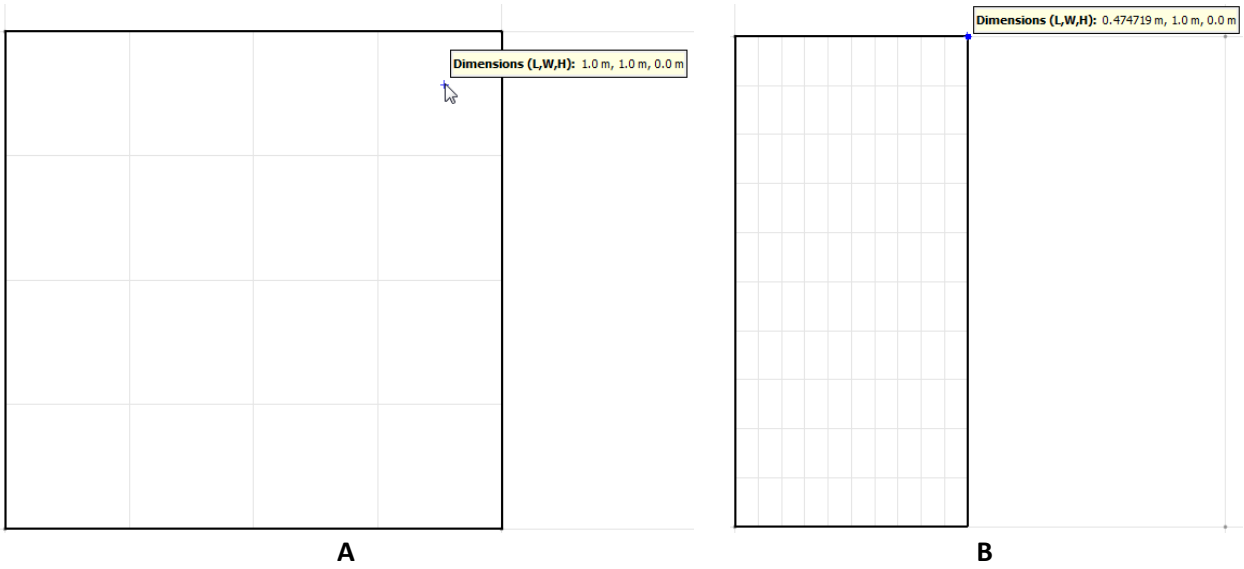


Figure 8.22. Two types of drawn meshes

To draw a solution mesh, perform the following:

1. Select the **Solution Mesh Tool** .
2. Use the **Click-drag Mode** or **Multi-click Mode** to draw two points defining the extents of the mesh.

Mesh Splitter Tool

Solution meshes can be easily split into two or more sub-meshes by using the **mesh splitter tool** . To split one or more meshes, perform the following:

1. Select the desired meshes to split either in the **navigation view** or by holding `g` on the keyboard while selecting a grid in the **2D** or **3D view**.
2. Select the **Mesh Splitter Tool** .
3. Move the cursor over one of the meshes. A preview slice will show where the meshes will be split as shown in Figure 8.23. By default the tool will choose the split axis based on the nearest boundary edge of a selected mesh. The split axis can be explicitly set through the right-click

menu, in the tool properties dialog, by click-dragging the left mouse button, or by pressing **CTRL** on the keyboard to cycle through the options.

4. Once the desired split location can be seen, left-click the mouse to set the location.
5. Repeat steps 3) and 4) to set more split locations. The meshes can be split along any number of axes in a single operation.
6. To perform the split, right-click in the **2D** or **3D view** and select **Finish**.
7. PyroSim will create a new mesh group for each selected mesh and place the resulting meshes in the appropriate groups.

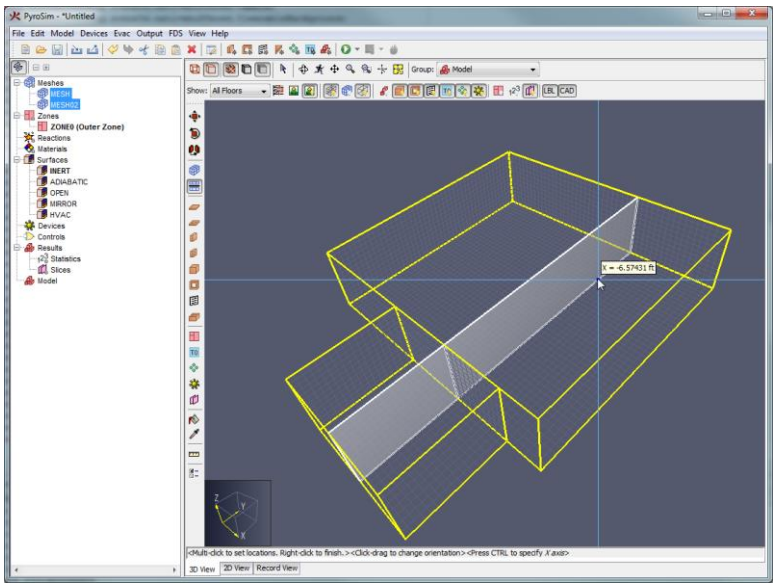


Figure 8.23. Meshes being split by the mesh splitter tool

Device Tool

PyroSim allows point devices to be drawn with the **device tool** (for more information on devices, see Chapter 144).

The device tool has the following properties:

- **[X,Y, or Z] Location:** The drawing plane on which to place the device. When the active **floor** is changed, this is set to the floor's **elevation** plus the floor's **wall height***.5.
- **Device Type:** The type of point device to draw. This can include sprinklers, smoke detectors, gas-phase devices, solid-phase devices, etc. Changing the **device type** also changes the other properties editable in the properties dialog. For more information on properties for each device type, see 0.

To draw a device, perform the following:

1. Select the **Device Tool** .
2. Click on the desired location.

NOTE: When drawing a device in the **3D View**, the device's location will snap differently depending on whether the device is a solid-phase device, such as one that measures wall temperature, or a gas-phase device, such as a smoke detector. If it is a solid-phase device, the device's location will snap directly to the point under the cursor if one is found. This makes it trivial to attach a solid-phase device to an obstruction. If the device is a gas-phase device, however, the device's location will be projected to the plane as set in the **device tool** properties as shown in Figure 8.24. This makes it easy to draw devices at a specific height above the floor. This behavior can be overridden for either type of device by selecting **Lock Z to [V]** (where **V** is the drawing plane) or **Lock Z to Snap Location** in the tool's right-click menu. **Lock Z to [V]** is the automatic behavior for gas-phase devices and **Lock Z to Snap Location** is the automatic behavior for solid-phase devices.

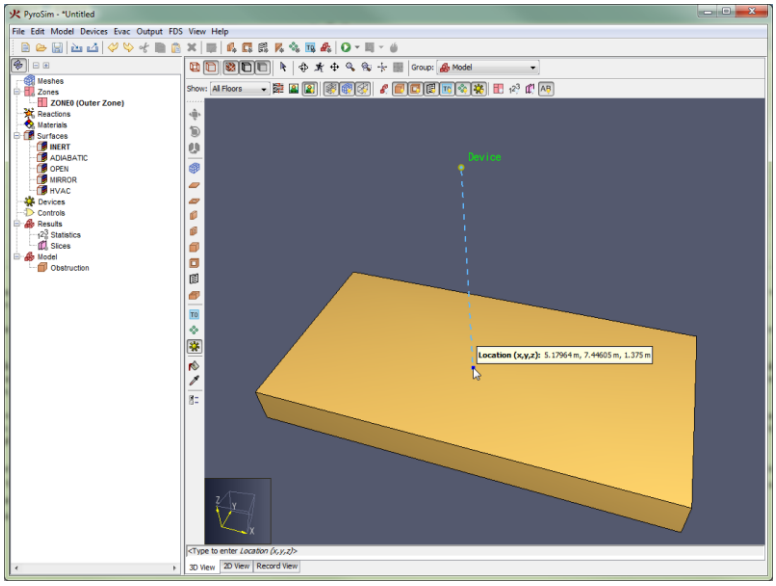


Figure 8.24. A gas-phase device being drawn in the 3D View

Planar Slice Tool

Planar slices, as discussed in Slices on page 116 can be drawn with the **planar slice tool** . To do so, perform the following:

1. Select the **Planar Slice Tool** .
2. In the tool properties dialog, select the desired quantity to record on the slice, the desired plane on which to draw the slice, and whether to include flow vectors.
3. Move the cursor to the desired slice location. A preview slice will be displayed as shown in Figure 8.25.[1] The slice plane can be changed through the right-click menu, by click-dragging the left mouse button, or by pressing **CTRL** on the keyboard to cycle through the options.
4. Left-click the mouse to create the slice.

[1] A planar slice can only be seen if the slice intersects a solution mesh.

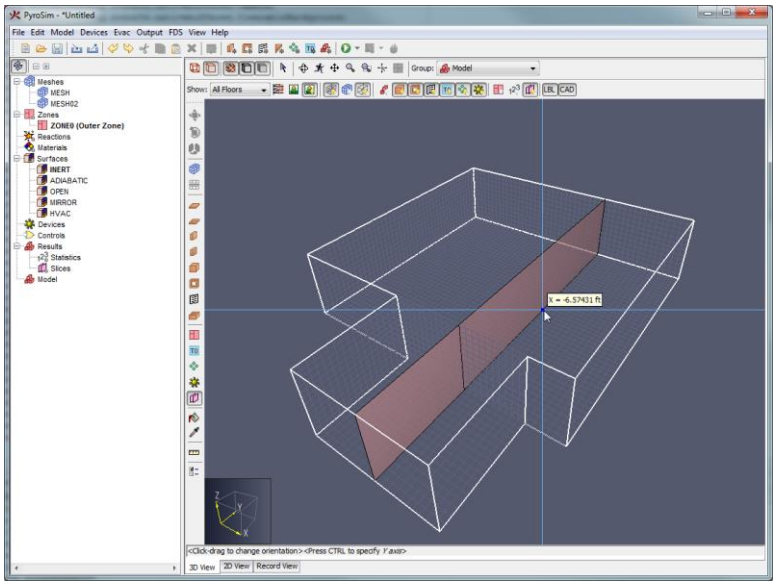


Figure 8.25. Drawing a slice with the planar slice tool

HVAC Node Tool

HVAC nodes as discussed in Chapter 15, can be drawn with the HVAC Node Tool . To do so, perform the following:

1. Select the **HVAC Node Tool** .
2. In the tool properties dialog, select the desired height of the node.
3. Move the cursor to the desired slice location.
4. Left-click the mouse to create the node.

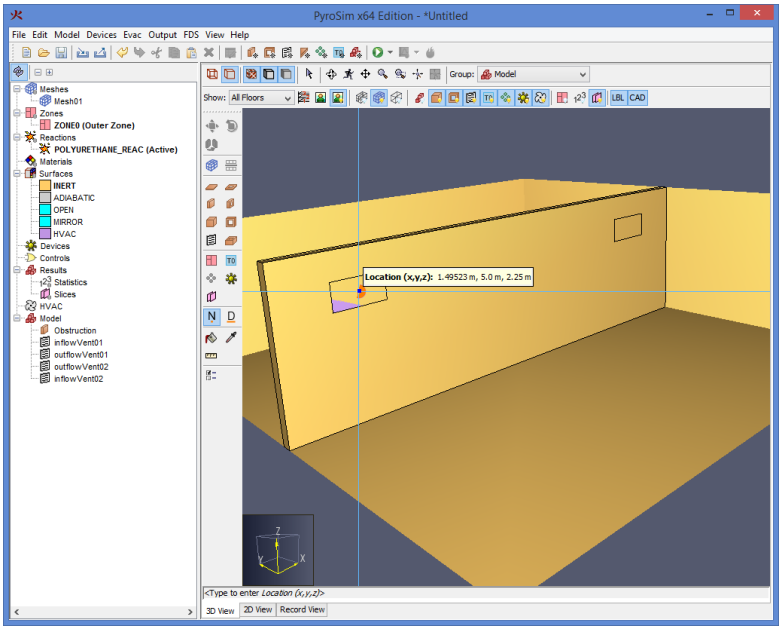


Figure 8.26. Using the HVAC Node drawing tool

HVAC Duct Tool

PyroSim allows HVAC ducts to be drawn with the HVAC Duct Tool D. To do so, perform the following:

1. Select the **HVAC Duct Tool** D.
2. Click to select the first HVAC Node. It is best to select the nodes in the direction air will flow through them.
3. After selecting the first node, a green line will appear from the node to the cursor, representing the duct. Complete the duct by selecting the second node.

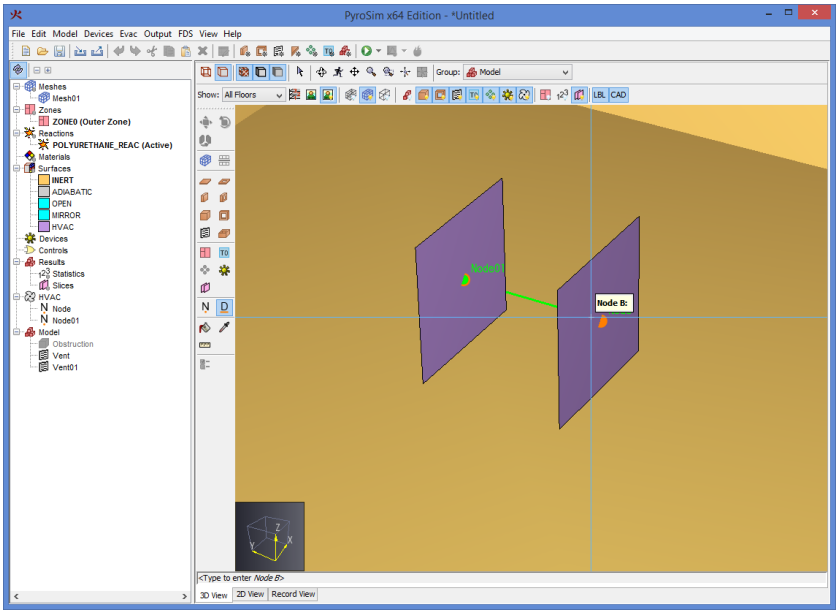


Figure 8.27. Drawing an HVAC Duct

Other Drawing Tools

PyroSim can also be used to draw **Init Regions** (page 126) with the **Init Region Tool** T0**, Particle Clouds** (page 99) with the **Particle Cloud Tool**, and **Zones** with the **Zone Tool**. These tools draw axis-aligned boxes, and so they behave similarly. They all have the following drawing properties:

- **[X,Y, or Z] Location:** The drawing plane for the tool. When the active **floor** is changed, this is set to the floor's **elevation**.
- **Height:** The depth of the drawn box in the current view. When the active **floor** is changed, this is set to the floor's **wall height**.

To draw one of these objects, perform the following:

1. Select the appropriate tool.
2. Use **Click-drag Mode** or **Multi-click Mode** to draw two points defining the extents of the object's box.

Editing Objects

Nearly all geometric objects can also be graphically edited in the **3D** or **2D View** with the **Select/Manipulate Tool** .

Editing is performed through an object's editing **handles**. **Handles** appear on an object either as a blue dot as shown in Figure 8.28 or a face with a different color. The dots indicate a point that can be moved in either two or three dimensions. A discolored face indicates that a face can be moved or extruded along a line.

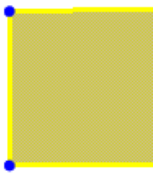


Figure 8.28. Editing handles on an object

To graphically edit an object, perform the following:

1. Select the **Select/Manipulate Tool** from the navigation toolbar.
2. Select one object to edit[1]. If the object can be graphically edited, blue editing handles will appear.
3. Hover the cursor over the desired handle. If the handle is a dot, it will turn yellow. If the handle is a face, the entire face will turn yellow as shown in Figure 8.29.
4. Move the handle using **Click-drag Mode** or **Multi-click Mode** to specify two points defining the movement vector.

[1] Only one object can be edited at a time. Editing handles will only appear if exactly one object is selected.

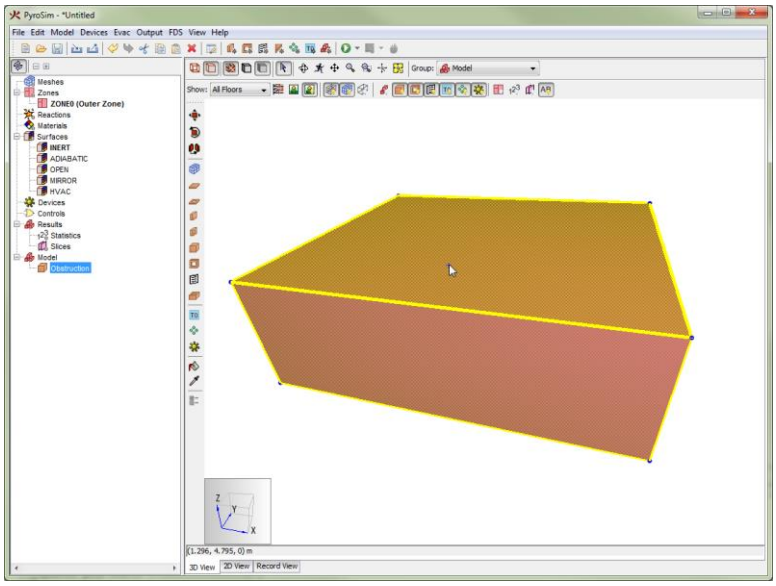


Figure 8.29. A highlighted face handle

Transforming Objects

PyroSim provides a variety of tools to transform geometry objects. With the transform tools, users can move, rotate, and mirror objects.

Copy Mode

Each tool has an alternate mode to copy the source objects with the transform. To toggle **copy mode** on/off, press the **CTRL** key on the keyboard. Alternately, choose **Copy Mode** or **Move Mode** from the tool's right-click menu. When using **copy** mode, the selected objects are copied and the copies are transformed.

Move Tool

This tool allows the user to move selected objects to a new location as shown in Figure 8.30. To use this tool, perform the following:

1. Select the desired objects to move from any of the views.
2. Select the **Move Tool** from the drawing toolbar.
3. Use **Click-drag Mode** or **Multi-click Mode** to draw two points defining the movement vector.

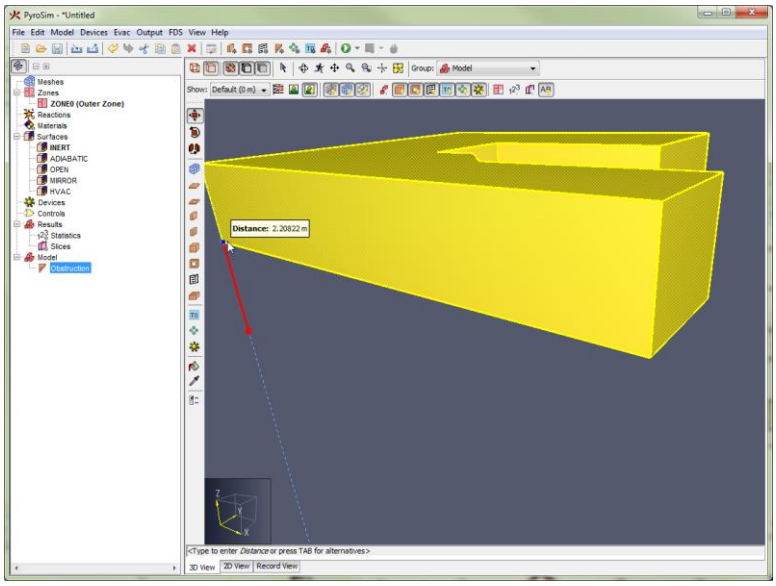


Figure 8.30. Moving an object using the Move Tool

Rotate Tool

This tool allows the user to rotate selected objects as shown in Figure 8.31[1]. To use this tool, perform the following:

1. Select the desired objects to rotate from any of the views.
2. Select the **Rotate Tool** from the drawing toolbar.
3. Single-click to specify the rotation center.[2]
4. Single-click another point to define a **reference vector** from the first point to the second point. This **reference vector** is what the angle is based from.
5. Single-click a third point to define the **angle vector** from the first point to the third point. The selected objects will be rotated by the angle between the **reference** and **angle** vectors.

The following defines the axis about which objects are rotated for each view:

- **3D View:** Z axis
- **Top View:** Z axis
- **Front View:** -Y axis
- **Side View:** -X axis

[1] Some objects, such as a solution mesh can only be rotated by 90 degree increments.
[2] The rotate tool can only be performed with **Multi-click Mode** since it requires three reference points.

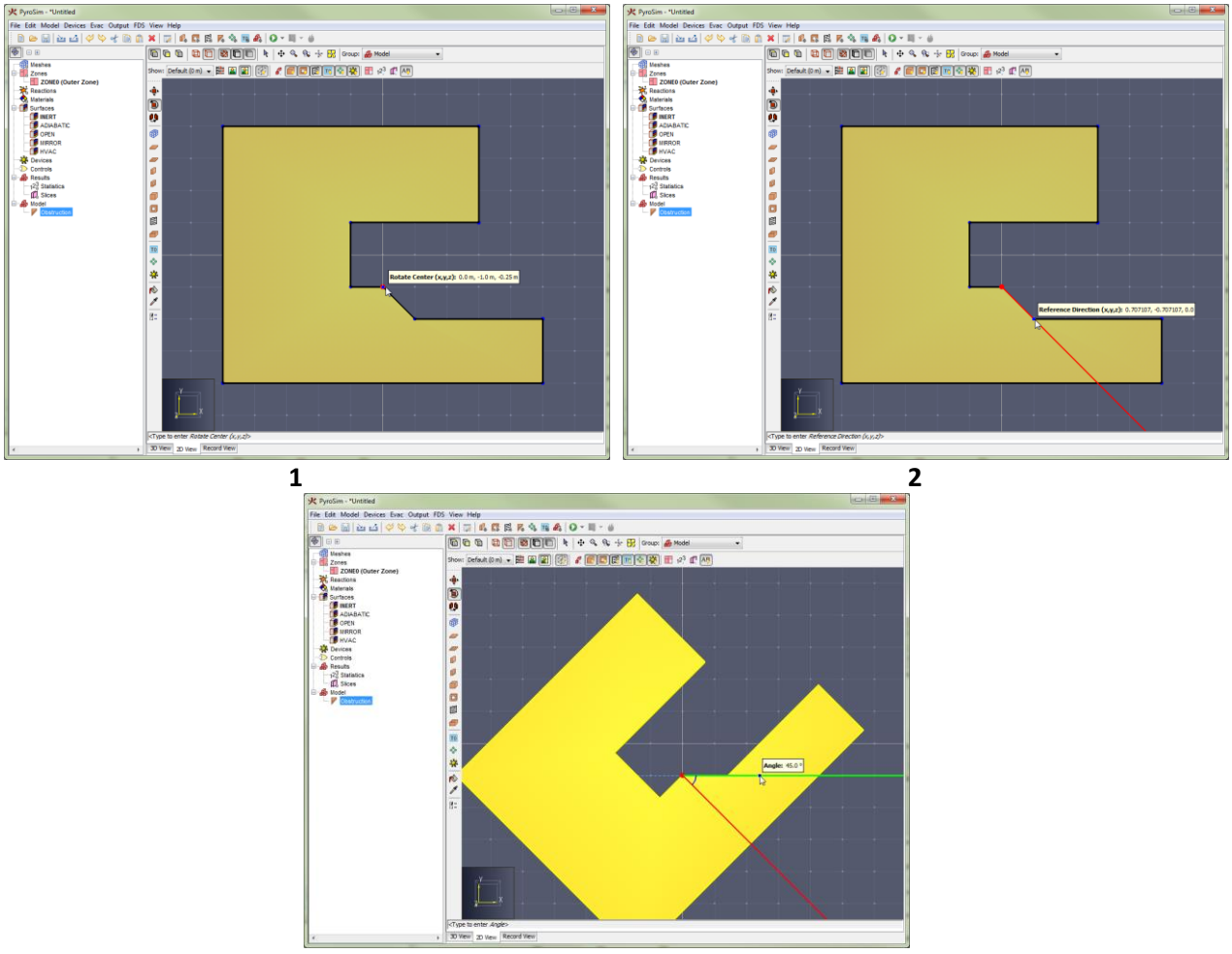


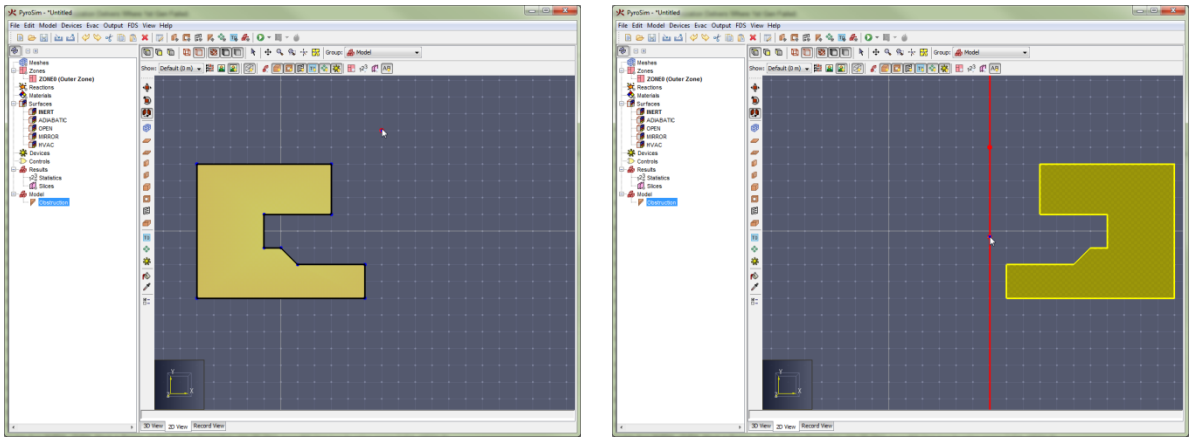
Figure 8.31. Rotating an object with the Rotate Tool

Mirror Tool

The mirror tool allows objects to be mirrored across a plane as shown in Figure 8.32[1]. To use the mirror tool, perform the following:

1. Select the desired objects to mirror from any view.
2. Select the **Mirror Tool** from the drawing toolbar.
3. Use **Click-drag Mode** or **Multi-click Mode** to define two points that create a plane about which the selected objects are mirrored.

[1] Some objects, such as solution meshes, can only be mirrored about an X, Y, or Z plane.



1 2

Figure 8.32. Mirroring an object using the Mirror Tool

Painting Obstructions and Vents

PyroSim provides tools to paint obstructions and vents with a surface and/or color. A user can also use a picking tool to pick the surface and/or color used to draw new obstructions or vents or to perform painting.

Paint Tool

A user can use the **Paint Tool** to paint the faces of obstructions and vents a specific surface and/or color.

The **Paint Tool** has the following tool properties:

- **Apply Surface:** Specifies whether to paint faces with **surfaces** and which surface to use.
- **Apply Color:** Specifies whether to paint faces with a specific color and which color to use.

These painting options can also be cycled by pressing **CTRL** on the keyboard or through the tool's right-click menu. The right-click menu also allows quick selection of a surface in the model or recently-used color.

To paint faces with the paint tool, perform the following:

1. Select the **Paint Tool** from the drawing toolbar.
2. Hover the cursor over the desired face to paint. If the face can be painted with the current surface/color, the face with highlight yellow as shown in Figure 8.33.
3. Either single-click the face to paint only that face, or click-drag the left mouse button to paint all faces the cursor moves over. Hold **SHIFT** on the keyboard while painting to paint all the faces of the object rather than just one face.

NOTE: A popup window will show which surface/color is currently being painted, but unlike other popup windows, this one is NOT editable.

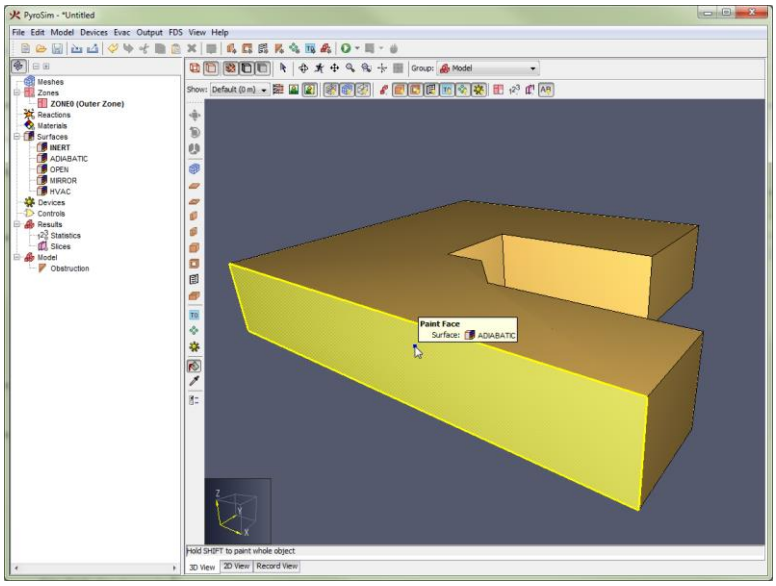


Figure 8.33. An obstruction face highlighted by the paint tool

Pick Tool

The **Pick Tool** can be used to pick the next color/surface to use when drawing or painting obstructions and vents.

The **Pick Tool** has the following tool properties:

- **Pick Surface:** Whether to pick the surface under the cursor
- **Pick Color:** Whether to pick the color under the cursor.

The pick options can also be toggled by pressing **CTRL** on the keyboard or from the tool's right-click menu.

To use the **Pick Tool**, perform the following:

1. Select the **Pick Tool** from the drawing toolbar.
2. Hover the cursor over the object from which the color/surface should be chosen. A popup window will show what that color/surface is.
3. Single-click the left mouse button to pick.

The chosen color/surface will then be applied to the surface or color properties for the obstruction, vent, mesh, and paint tools.

Measuring Length/Distance

PyroSim provides a **measure tool** to measure distances in the model. To measure a distance, perform the following:

1. Select the **Measure Tool** **.**

2. Either click several points to measure the total distance along a path or click-drag to measure a single distance between two points. As the cursor is moved, the current cursor location, total path length, and length of the most recent segment are displayed as shown in Figure 8.34.
3. To copy the distance, right-click and from the menu, choose **Copy total distance to clipboard**. This will make the total distance measurement available on the clipboard so it can be pasted into a field or any other paste location. The text that is copied to the clipboard includes both the full precision value and the current display length unit. For example, it might copy "3.2808399 ft" if working in the English unit system.

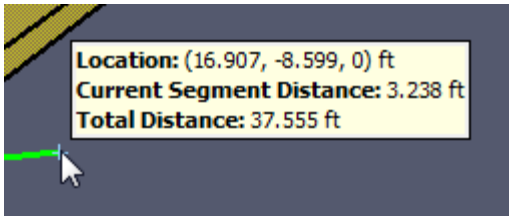


Figure 8.34. Information displayed by the measure tool

NOTE: When working in the **3D View**, all distances are true to the 3D snapped locations of the cursor. In the **2D Views**, however, the cursor locations are projected to a single plane parallel to the view before determining the distance. This means that in the **Top View**, all Z distances are ignored. In the **Front View**, all Y distances are ignored. In the **Side View**, all X distances are ignored.

Chapter 9. Creating Complex Geometry

This chapter provides guidance on using the geometry tools available in PyroSim to create several geometric shapes that often appear in building models. The ability to sketch in different planes, copy, replicate, drag, scale, and rotate objects can greatly simplify the tasks of geometry creation.

Curved Walls

While PyroSim's tools do not explicitly produce curved walls, they can approximate them using any of the following techniques:

- Draw the wall using several straight wall segments.
- Draw the wall using individual blocks.
- Rotate a single object to produce the desired arc.

In all of the following examples, we will use a background image as a pattern to draw against. While this is not required, it makes creating curved surfaces much easier and one of the strengths of PyroSim is that it allows you to sketch geometry directly on top of building design images. The background image we will be using is shown in Figure 9.1.

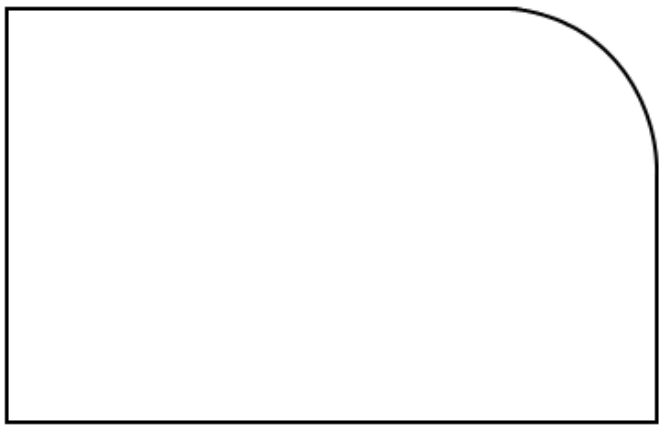


Figure 9.1. Background image used for all curved wall examples

For simplicity, we will assume that horizontal distance across the entire image is 50 feet, and we will place the origin of the model at the lower-left corner of the room shown in the image. The brightness of the image will be set to 50%. The **Configure Background Image** dialog shown in Figure 9.2 illustrates these settings.

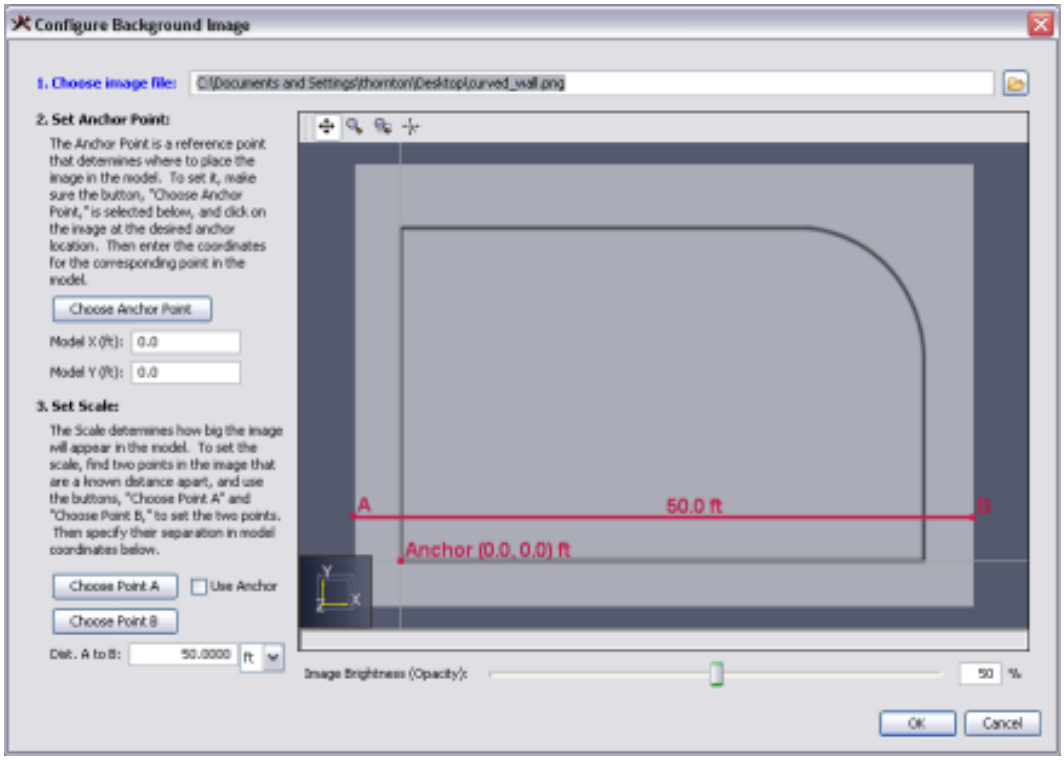

Configure Background Image
1. Choose image file:
C:\Documents and Settings\thornton\Desktop\curved_wall.png
2. Set Anchor Point:
The Anchor Point is a reference point that determines where to place the image in the model. To set it, make sure the button, "Choose Anchor Point," is selected below, and click on the image at the desired anchor location. Then enter the coordinates for the corresponding point in the model.
Choose Anchor Point
Model X (ft): 0.0
Model Y (ft): 0.0
3. Set Scale:
The Scale determines how big the image will appear in the model. To set the scale, find two points in the image that are a known distance apart, and use the buttons, "Choose Point A" and "Choose Point B," to set the two points. Then specify their separation in model coordinates below.
Choose Point A
Use Anchor
Choose Point B
Dist. A to B: 50.0000 ft
A
50.0 ft
B
Anchor (0.0, 0.0) ft
Image Brightness (Opacity): 50 %
OK
Cancel

Figure 9.2. Background image settings for curved wall examples

Using the Wall Tool

To create a curved wall section from wall segments, you can follow these steps:

1. Click the **2D View** tab, and select the **Draw Wall Obstruction** tool.
2. Turn off grid snapping. In the **View** menu, click **Disable Grid Snapping**.
3. Position the cursor at the beginning of the curve where you want to place the first wall segment.
4. Use **Multi-click Mode** to click several points along the curve. More points will create a smoother curve.
5. Right-click in the **2D View** and select **Finish** to finish drawing the wall.

This is the fastest way to create smooth curves in PyroSim. PyroSim will convert the curved walls to blocks before running the FDS simulation. While smaller segments will make the wall look better in PyroSim, placement of obstructions generated for FDS depends on the resolution of your mesh. Three different versions of a curved wall created with this technique are shown in Figure 9.3.

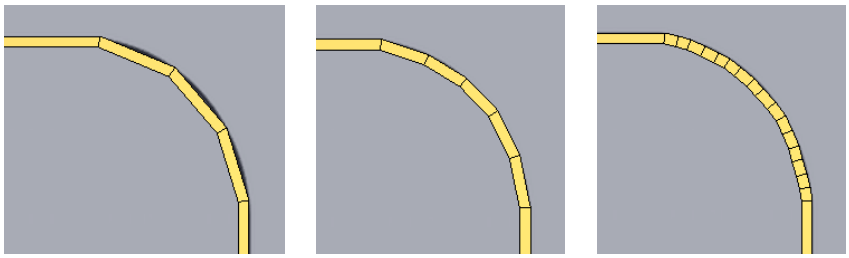


Figure 9.3. A curved wall drawn with three different segment lengths

Using extremely short line segments will probably not be of any benefit unless you also use very small mesh cells.

Using the Block Tool

To create a curved wall section from blocks, you can follow these steps:

1. Create a mesh. This example uses a `50.0 ft x 50.0 ft` mesh with `1 ft` mesh cells.
2. Click the **2D View** tab, and select the **Block Obstruction Tool** .
3. Turn grid snapping on. If snapping is off: in the **View** menu, click **Snap to Model Grid**.
4. Click each cell along the curved wall to place the necessary blocks.

This technique forces you to convert the curve to blocks manually, but the advantage is you know exactly what geometry will be generated for FDS. If you have a high resolution mesh, it may be useful to drag the mouse and "paint" the curve rather than clicking individual blocks. The example curved wall is shown in Figure 9.4.

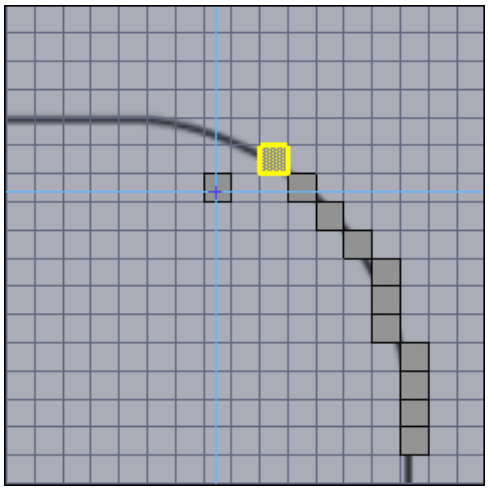


Figure 9.4. A curved wall drawn using individual blocks

Rotating an Object

To create curved objects using the rotation technique, you must place an initial segment, then perform a rotate-copy operation about the center point of your desired curve. This process is illustrated in the following steps:

Click the **2D View** tab, and select the **Wall Obstruction Tool** .

1. Turn off grid snapping. If snapping is on: in the **View** menu, click **Disable Grid Snapping**.
2. Create an initial wall segment somewhere on the curve.
3. In the **Model** menu, click **Rotate...**
4. Select the **Copy** mode.
5. Specify the necessary parameters for the rotation operation. In this example, the **Number of Copies** is `15`, the **Angle** is `6.0` degrees, and the **Base Point** is: x=`32.0` feet, y=`16.5` feet.
6. Click **Preview** to verify that the settings are correct, then click **OK**.

The curve for this example is shown in Figure 9.5.

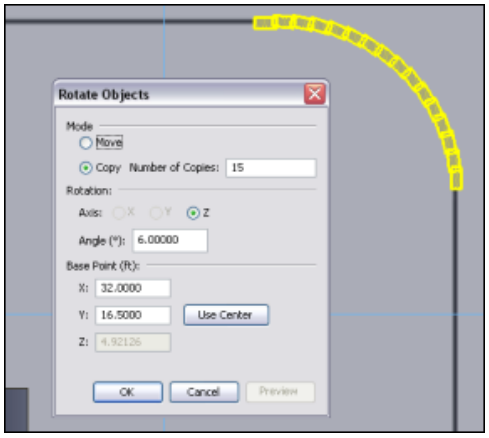


Figure 9.5. A curved wall drawn using the rotate technique

If we would have created 60 copies instead of 15 this procedure would have created a cylinder. While complicated, the rotation approach is the most effective at creating complex symmetrical geometry.

Trusses

Trusses can be created by drawing a single truss out of slab obstructions and slab holes, then replicating that truss as many times as needed as shown in Figure 9.6. The following steps show how to create the trusses for an example roof.

1. Click the **2D View** tab
2. On the toolbar, click the **Front View** button.
3. On the drawing toolbar, select the **Slab Obstruction Tool**.
4. On the drawing toolbar, click the **Tool Properties** button, and set the **thickness** to the desired thickness of the truss.
5. Draw a triangle representing the outer boundary of the truss.
6. On the toolbar, select the **Slab Hole Tool** .
7. Draw a triangular hole in the left half of the truss.
8. Select the newly created hole.
9. Select the **Mirror Tool**.
10. Right click in the **2D View** and select **Copy Mode**.
11. Draw a line from the top vertex of the outer obstruction to the center of the bottom to define the mirror plane.
12. Select the entire truss, including the outer obstruction and two holes.
13. Open the **Model** menu, and click **Copy/Move...**.
14. In the **Translate** dialog: select **Copy**, set **Number of Copies** to `4`, set **Offset** to be `2.0` meters along the Y axis, and click **OK**.

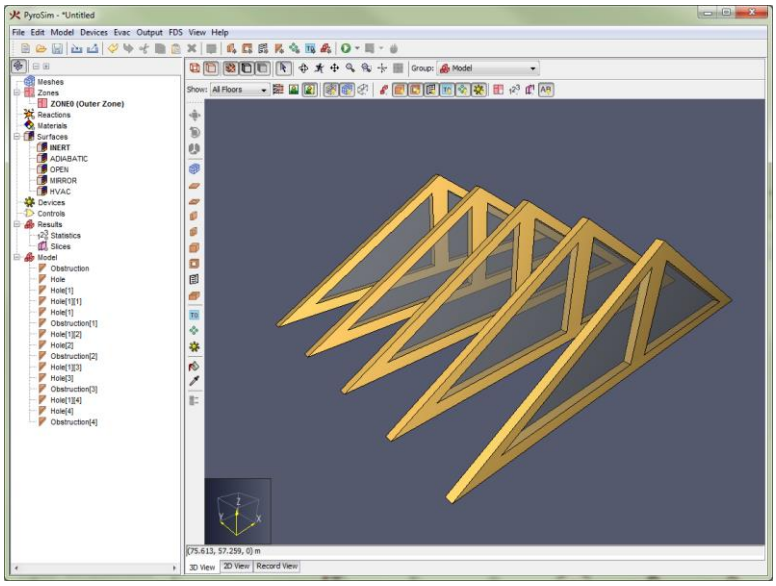


Figure 9.6. Trusses created using PyroSim tools

Roofs

You can quickly add a roof to the model using the **Slab Obstruction Tool** . The following steps show how to add a roof to the previous truss example.

1. Click the **2D View** tab.
2. On the view toolbar, select the **front** view .
3. On the drawing toolbar, select the **Slab Obstruction Tool** .
4. Draw the profile of the roof as one polygon so that it touches the trusses. Click the top point, then bottom point, and the top point once more to create the slab.
5. On the view toolbar, select the **top** view .
6. On the navigation toolbar, select the **Select/Manipulate Tool** .
7. Click to drag the leading edge of the slab to the opposite truss.
8. Single-click to end manipulation.
9. Repeat steps 7-9 for the back edge of the roof slab to line it up with the back of the last truss.

The resulting roof section is shown in Figure 9.7.

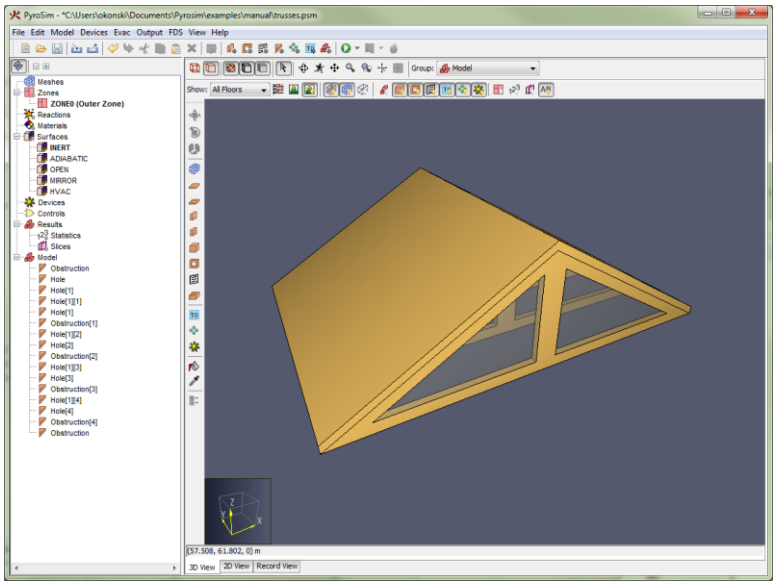


Figure 9.7. A roof created with the slab obstruction tool

The extruded polygon tool can be used to create obstructions with any number of boundary points (triangles, quads, etc).

Stairs

Users can create simple stairways by placing the initial stair, then using the translate-copy operation. This section will present a simple example to illustrate the approach.

We will create a 10 step stairway. Each step will have a 7 inch rise (0.58 feet), and a 10 inch (0.83 feet) run. The stairway itself will be 24 inches (2.0 feet) wide. To keep things as simple as possible, we will construct the stairway in an empty model.

1. On the **Model** menu, click **New Obstruction...**
2. In the **Obstruction Properties** dialog, specify the min point as `(0.0, 0.0, 0.0)` and the max point as `(2.0, 0.83, 0.58)`.
3. On the **Model** menu, click **Copy/Move...**.
4. In the **Translate** dialog, select **Copy**, set the **Number of Copies** to `9`, set the **Offset** to `(0.0, 0.83, 0.58)`, and click **OK**.

The stairway generated in this example is shown in Figure 9.8.

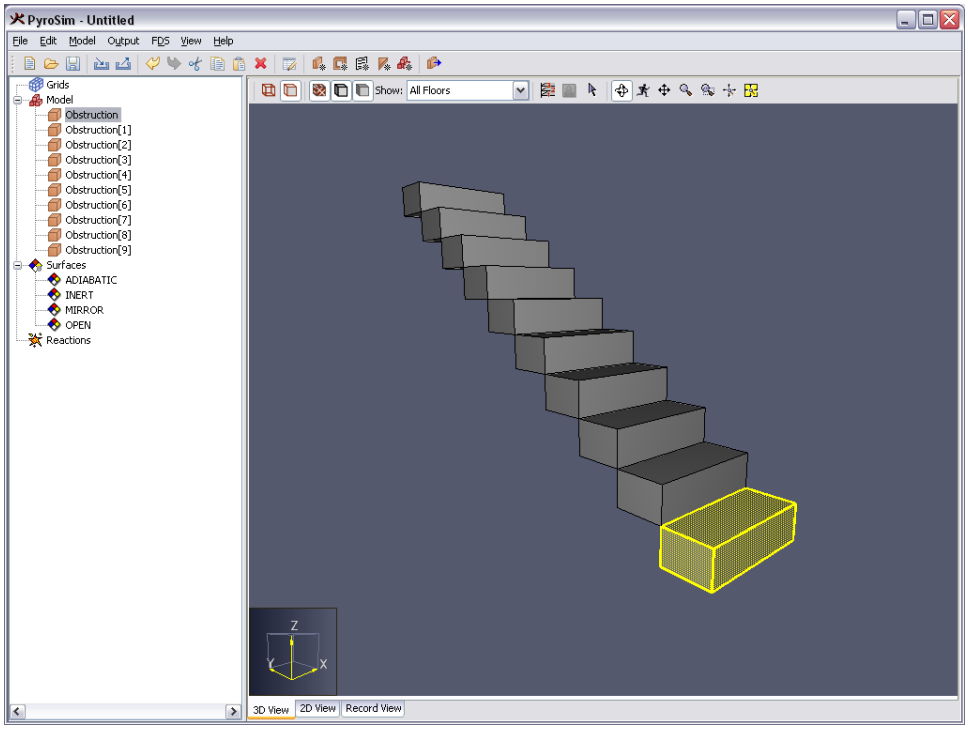


Figure 9.8. A stairway created with the replicate tool

Chapter 10. Working with Geometry Objects

Selection

PyroSim relies heavily on the idea of selected objects. For almost all operations, the user first selects an object(s) and then changes the selected object(s). The **Selection Tool** is used to select objects.

- A left mouse click on an object in any view will select it.
- Holding **Ctrl** while clicking will toggle the item in the selection, adding previously unselected items and removing previously selected items.
- Holding **Alt** while clicking an object in the 3D View or the 2D View will select the entire group that the object belongs to.
- In the Navigation View, a range of objects can be selected by clicking the first object, then holding **Shift** while clicking the last object.
- In the 2D and 3D Views, multiple objects can be selected with the **Selection Tool** and a click-drag motion to define a selection box.

Once objects have been selected, the user can modify the object using the menus.

Selection can be made in any of the views using the **Selection** tool. Multiple objects can be selected using the **Ctrl** key or click and drag to define a box. In the Navigation View, the **Shift** key can be used to select a consecutive list of objects.

Context Menus

A right-click on a selection displays a context menu. This menu includes the most common options for working with the object. The user may also right-click on individual objects for immediate display of the context menu.

Undo/Redo

All geometric changes to the model can be undone and redone using the Undo and Redo buttons, as well as **Ctrl+Z** and **Ctrl+Y**, respectively.

Copy/Paste

Select an object to copy, then either use **Ctrl+C** or **Edit->Copy** to copy. Alternately, right-click on an object to display the context menu with **Copy**.

Either use **Ctrl+V** or **Edit->Paste** to paste a copy of the object. Alternately, right-click on an object to display the context menu with **Paste**.

Copy/Paste from Other Models

By running two instances of PyroSim, you can copy objects from one model and paste them into a second model. If the copied objects rely on other properties, such as surfaces, that are not included in the second model, these properties will be pasted into the model when the objects are pasted.

Copy/Paste from Text Files

Copy/paste can also be performed to and from text files. For example the user can select an object in PyroSim, open a text file, and paste the object. The text FDS representations of the object and dependent properties will be pasted. Alternatively, the user can copy the text from an FDS file and paste into PyroSim (the 3D View, 2D View, or Navigation View). The object will be added to the PyroSim model. An error message will be received if the pasted object depends on data that is not available in the PyroSim model. The user will then need to paste that information (such as surface properties) first before pasting the geometric object.

Double-Click to Edit

Double-clicking on an object opens the appropriate dialog for editing the object properties.

Translate and Copy Dialog

The Translate dialog can be used to both move an object and to create copies of an object, each offset in space. To move an object in this manner, perform the following:

1. Right click the object(s) to move.
2. From the right-click menu, select **Copy/Move…**.
3. This will show the Translate dialog, Figure 10.1.

The **Mode** selects either the option to move only the selected object or to create copies of the object and move them. The **Offset** parameters indicate the increment to move or offset the copies.

To preview the changes without applying them, click **Preview**. To apply the changes and close the dialog, click **OK**. To cancel the changes instead, click **Cancel**.

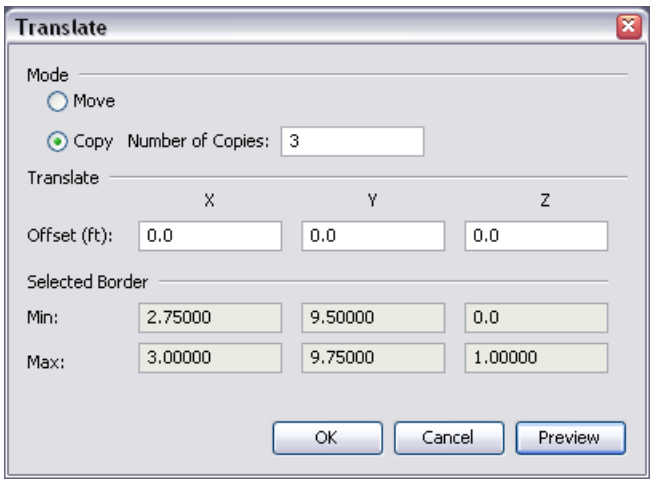


Figure 10.1. The translate dialog

Mirror and Copy Dialog

The Mirror dialog can be used to mirror an object about a plane or planes. To mirror an object in this manner, perform the following:

1. Right click the object(s) to mirror.
2. From the right-click menu, select **Mirror…**.
3. This will show the Mirror dialog as in Figure 10.2.

The **Mode** selects either the option to mirror only the selected object or to create a mirrored copy of the object. The **Mirror Plane(s)** define planes normal to the X, Y, and Z axes about which the object will be mirrored. The **Use Center** button can be used to fill the Mirror Plane data with the center coordinates of the selected objects.

To preview the changes without applying them, click **Preview**. To apply the changes and close the dialog, click **OK**. To cancel the changes instead, click **Cancel**.

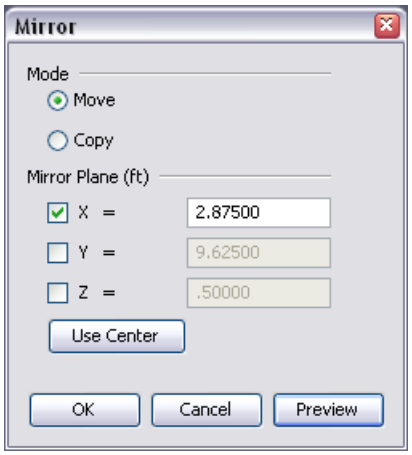


Figure 10.2. The mirror dialog

Scale and Copy Dialog

The Scale dialog can be used to change the size of an object. To scale an object, perform the following:

1. Right click the object(s) to scale.
2. From the right-click menu, select **Scale…**.
3. This will show the Scale dialog as in Figure 10.3.

The **Mode** selects either the option to scale only the selected object or to create multiple scaled copies of the object. The **Scale** values define the scale factors in the X, Y, and Z directions. The **Base Point** defines the point about which the scaling will be performed. The **Use Center** button can be used to fill the Base Point data with the center coordinates of the selected objects.

To preview the changes without applying them, click **Preview**. To apply the changes and close the dialog, click **OK**. To cancel the changes instead, click **Cancel**.

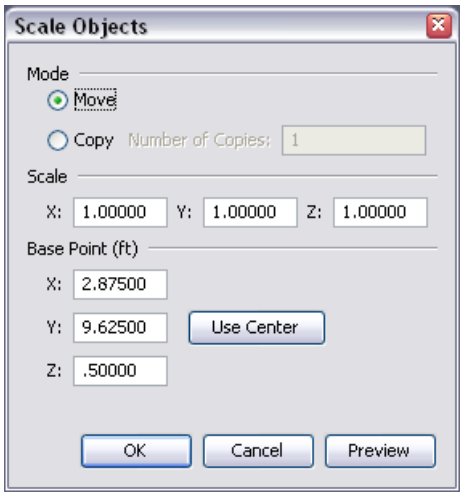


Figure 10.3. The scale dialog being used to scale an object

Rotate and Copy Dialog

The Rotate dialog can be used to rotate an object. To rotate an object, perform the following:

1. Right click the object(s) to rotate.
2. From the right-click menu, select **Rotate**…
3. This will show the Rotate dialog as in Figure 10.4.

The **Mode** selects either the option to rotate only the selected object or to create multiple rotated copies of the object. The **Rotation** values allow the user to select the axis about which the rotation will be made and the angle is the rotation angle (counter-clockwise is positive). The **Base Point** defines the point about which the rotation will be performed. The **Use Center** button can be used to fill the Base Point data with the center coordinates of the selected objects.

To preview the changes without applying them, click **Preview**. To apply the changes and close the dialog, click **OK**. To cancel the changes instead, click **Cancel**.

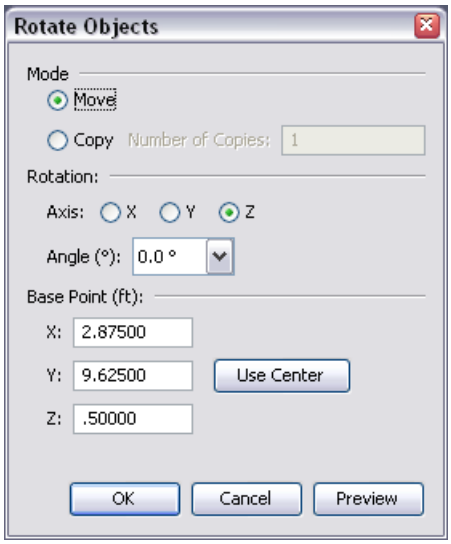


Figure 10.4. The rotate dialog being used to rotate an object

Object Visibility

Often it is desirable to turn off the display of selected objects, for example, to hide a roof of a building in order to visualize the interior. In any of the views, right-click on a selection to obtain the following options:

- Hide object(s) - This turns off the display of the selected object(s).
- Show object(s) - This turns on the display of the selected object(s).
- Filter object(s) - This turns off the display of all objects except the selection(s).
- Show all objects - Turns on the display of all objects.

Chapter 11. Species

Species may be used for a number of different purposes in a PyroSim model. They can be used as the reactive species for a combustion model, as a non-reactive species in a simulation of air flow, or simply as a component of a more complex species.

Species can be managed by opening the Model menu and selecting Edit Extra Species…. To create a new species, select New. You may select one of the predefined species provided by FDS, or create a custom species. You may also specify that the species is either Primitive or Lumped.

Primitive Species

Primitive species can be tracked individually, or as a component of a more complex lumped species.

To edit primitive species:

1. On the **Model** menu, click **Edit Extra Species…**.
2. On the **Primitive** tab, set the **Molecular Weight** (MW) of the species. You may also specify the chemical formula for the reaction if you plan to use the species in a combustion reaction. The **Primitive** tab for the predefined Extra Species, "WATER VAPOR".
3. On the **Vis/Dif** tab, you can specify the **Diffusivity** (DIFFUSIVITY) or **Viscosity** (VISCOSITY) for the species. These can either be fixed, or specified as ramps using the **Custom** option in the respective combo boxes. You can also specify the Lennard-Jones Parameters (SIGMALJ, EPSILONKLJ).
4. Some species need to absorb or emit thermal radiation. In this case, you may specify a **Radcal Surrogate** on the **Radiation** tab. The species will use the absorbing properties of the specified surrogate. It is best to use a surrogate with molecular properties similar to the custom species.
5. To invoke the aerosol deposition model of FDS, select the **Aerosol** (AEROSOL) checkbox on the **Soot** tab. If the deposition model is used, you must also specify Density (Solid) (DENSITY_SOLID), **Conductivity (Solid)** (CONDUCTIVITY_SOLID), and **Mean Diameter** (MEAN_DIAMETER).
6. The **Liquid** tab is used to define the thermal properties of evaporating liquid droplets. Note that these variables are only used when a particle evaporates. For more information about specifying an evaporating liquid droplet, see Chapter 13.
7. The **Gas** tab is used to set parameters related to the enthalpy of the gas species. The enthalpy is specified by a combination of **Specific Heat** (SPECIFIC_HEAT), **Reference Temperature** (REFERENCE_TEMPERATURE), and **Reference Enthalpy** (REFERENCE_ENTHALPY). For more information on how enthalpy is calculated, refer to the FDS User Manual (McGrattan, et al., 2013).

Lumped Species

Species mixtures can be defined as a mixture of any number of primitive species. Because all species in the simulation must be tracked by a transport equation, a lumped species can be used to save on simulation time.

To create a lumped species:

1. On the **Model** menu, click **Edit Extra Species…**.
2. Click **New**…
3. Select the **Lumped** option.
4. Click **OK**…

To edit the lumped species:

1. On the **Model** menu, click **Edit Extra Species…**.
2. Lumped species cannot be assigned fixed values like a primitive species can. Instead, the simulator derives this information from the combination of all the individual components of the species. In the **Lumped** panel, you can specify the composition as either a **Mass Fraction** or a **Volume Fraction**. In either case, a table of all declared Primitive Species and their fractional composition is displayed on the panel.

When using lumped species, it is recommended that certain actions be taken to reduce the complexity of the simulation. To save on simulation time:

1. On the **Model** menu, click **Edit Extra Species…**.
2. If a primitive species is to be used only as a component of a lumped species, select that species.
3. Click the **Advanced** panel.
4. Add the line LUMPED_COMPONENT_ONLY = .TRUE. to the list of **Additional Fields**. For more information about **Advanced** FDS parameters, see Chapter 21.
5. Click **OK**…

Doing this check for all primitive species will reduce the number of transport equations solved by the simulator, and save significant time on the simulation.

Chapter 12. Reactions

This chapter provides an overview of how to specify combustion (the reaction of fuel vapor and oxygen) using PyroSim. A more detailed discussion of this topic is provided in the Fire Dynamics Simulator User's Guide (McGrattan, et al., 2013) and the Fire Dynamics Simulator Technical Reference Guide (McGrattan, et al., 2013).

As described in the Fire Dynamics Simulator User's Guide (McGrattan, et al., 2005), a common source of confusion in FDS is the distinction between gas phase *combustion* and solid phase *pyrolysis*. The former refers to the reaction of fuel vapor and oxygen; the latter the generation of fuel vapor at a solid or liquid surface. In an FDS fire simulation, there is only one gaseous fuel that acts as a surrogate for all the potential fuel sources. The reaction is defined using the *Edit Reactions* dialog in PyroSim.

The PyroSim interface supports only the *single-step, mixing controlled combustion* model to account for the evolution of the fuel gas from its surface of origin through the combustion process. The alternative provided in FDS6 is the *finite-rate approach*, where all of the individual gas species involved in the combustion process are defined and tracked individually. This finite-rate approach is recommended only for Direct Numerical Simulation. The mixture fraction model is the most frequently used approach.

Mixture Fraction Combustion

In FDS, there are two ways of designating a fire: the first is to specify a Heat Release Rate Per Unit Area (HRRPUA) as part of a surface, the other is to specify a HEAT_OF_REACTION, along with other thermal parameters, as part of a material. In both cases, the mixture fraction combustion model is used.

The heat release rate approach is the simplest way to specify a fire. All that needs to be done is create a burner surface with the desired heat release rate (see Chapter 6). If no other reaction is specified, propane will be used as the surrogate fuel. If a reaction is specified, that reaction will be used to calculate the combustion products.

In the mixture fraction model, the reaction is assumed to be of the form:

$$C_xH_yO_zN_v + \nu_{O_2}\, O_2 \rightarrow \nu_{CO_2}\, CO_2 + \nu_{H_2O}\, H_2O + \nu_{CO}\, CO + \nu_S\, \text{Soot} + \nu_{N_2}\, N_2$$

Figure 12.1. Reaction equation

The user specifies the chemical formula of the fuel along with the yields of CO, soot, and H2, and the amount of hydrogen in the soot. For completeness you can also specify the N2 content of the fuel.

To edit a reaction:

1. On the **Model** menu, click **Edit Reactions...**.
2. On the **Fuel** tab, you may enter either the number of atoms in the reaction equation, or specify the reacting species. By default, PyroSim adds a custom species called REAC_FUEL to serve as the default fuel **Species**. If the species is not one defined by FDS, then the fuel composition must

be defined as well. The input for a polyurethane reaction described in the SFPE Handbook, is shown in Figure 12.2.

3. On the **Fire Suppression** tab, you can enable fire suppression and then enter values for the **Critical Flame Temperature** (CRITICAL_FLAME_TEMPERATURE) and the **Automatic Ignition Temperature** (AUTO_IGNITION_TEMPERATURE). The panel is shown in Figure 12.3.
4. On the **Byproducts** tab, you can select either **Specify release per unit mass oxygen** (EPUMO2) or **specify heat of combustion** (HEAT_OF_COMBUSTION). You can also specify the CO yield, H2 yield, and Soot yield. The values used for the polyurethane reaction are shown in Figure 12.4.

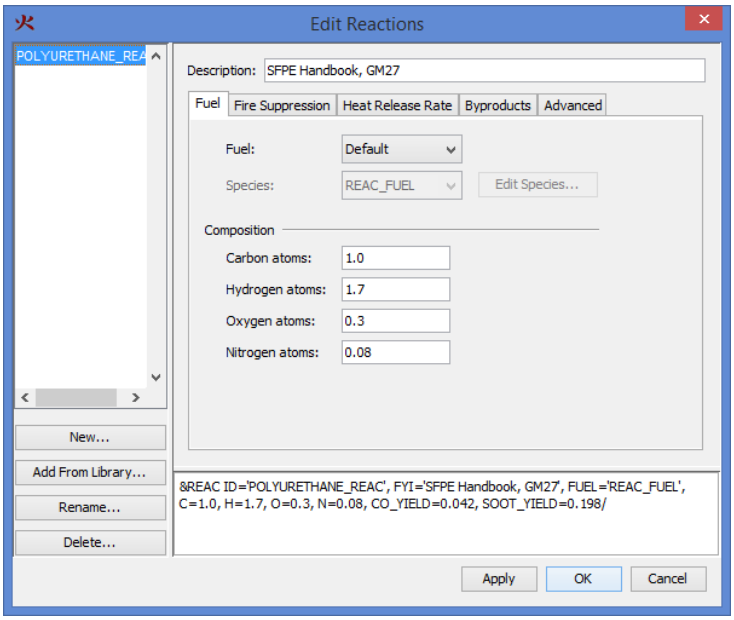


Figure 12.2. Fuel panel of the Edit Reactions dialog

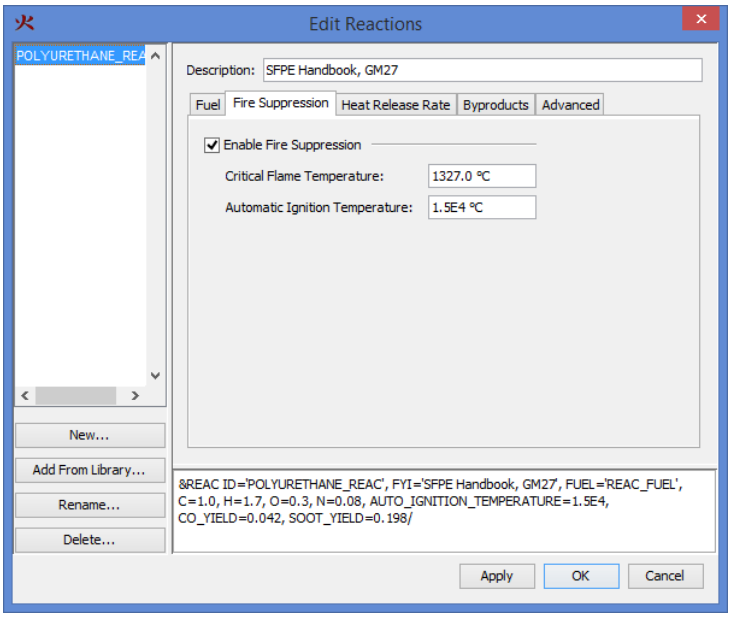


Figure 12.3. Fire Suppression panel of the Edit Reactions dialog

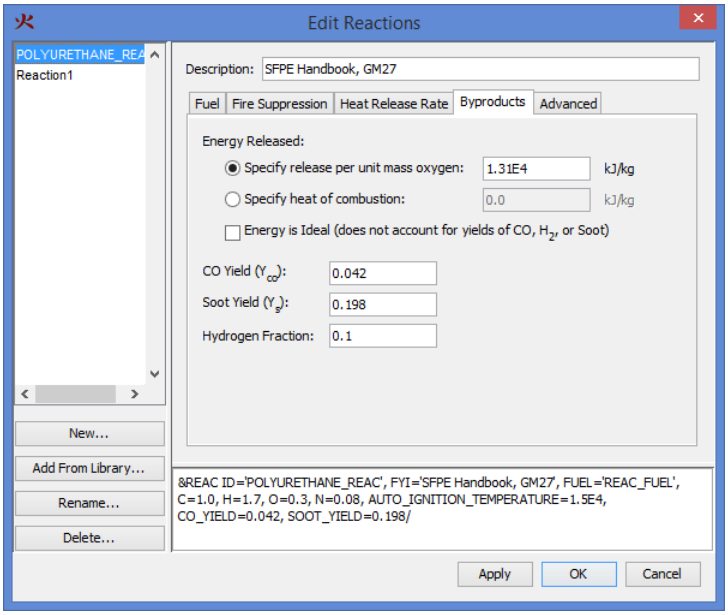


Figure 12.4. Byproducts panel of the Edit Reactions dialog

Custom Smoke

PyroSim supports the custom smoke features available in FDS. To create custom smoke, first define an extra species with the desired mass extinction coefficient. This “smoke” species can then be injected into the domain like any other extra species. Smoke detectors can also detect this smoke species if it is selected under the smoke detector’s model, which can be edited under **Edit Smoke Detector Models...** on the **Devices** menu. Finally, if Smokeview should track this species as smoke, go to the **FDS** menu, select **Simulation Parameters...**, and on the **Output** tab select the mass fraction quantity of the desired

species from the **Smoke Quantity** drop-down box. Note that in addition to specifying the mass fraction of an extra species, the mass fraction of any mixture fraction species can also be selected for smoke display, including the mass fraction of oxygen, water vapor, and the **other** species specified in the gas-phase reaction.

Chapter 13. Particles

PyroSim supports four types of particles: massless tracers, water droplets, fuel droplets, and a generic particle that provides the same options as water droplets.

Massless Tracers

Massless tracer particles can be used to track air flow within a simulation. They can be used with the particle injection feature of the **Burner**, **Heater/Cooler**, **Blower**, and **Layered** surface types. They can also be used in particle clouds.

By default, PyroSim provides a black, massless tracer particle called `Tracer`. To use a custom tracer particle in your simulation, you can modify the parameters of this default particle to suit your needs, or you can create a new particle. To create a new massless tracer particle:

1. On the **Model** menu, click **Edit Particles...**
2. Click **New...**
3. In the **Particle Type** box, select **Massless Tracer**
4. Click **OK**

You can now edit the particle properties.

Parameter	Description
Duration	The amount of time a droplet of this type will remain in the simulation.
Sampling Factor	Sampling factor for the particle output file. A value of `-1` uses the FDS default value for this property. Set to an integer greater than `1` to reduce the size of particle output.

Liquid Droplets

Evaporating liquid droplets can be used with sprinkler spray models and nozzles to customize the spray. They can also be used in particle clouds and surface types that support particle injection.

To specify a liquid droplet, you must specify a species. This can be one of the predefined species recognized in Table 11.1 of the FDS User's Guide (McGrattan, et al., 2013), or any other user defined species. If the species is not predefined, it is important to specify the liquid properties of the species.

To edit the liquid properties of the species:

1. On the **Model** menu, click **Edit Extra Species...**
2. Select the primitive species
3. Select the **Liquid** tab
4. Specify the fields below.
5. Click **OK**

Liquid Species:

Parameter	Description
Specific Heat	The droplet specific heat.
Density	The droplet density.
Vaporization Temperature	The droplet liquid boiling temperature.
Melting Temperature	The droplet melting/freezing temperature.
Heat of Vaporization	The droplet latent heat of vaporization.
Enthalpy of Formation	The heat of formation of the gaseous species.
H-V Reference Temperature	The temperature associated with the Heat of Vaporization

Coloring:

Parameter	Description
Default	Select to FDS to select a color for this particle.
Specify	Select to choose a custom particle color.
By Droplet Property	Select this option to choose one or more scalar quantities that will be used to color this particle in Smokeview.

Size Distribution:

Parameter	Description
Median Diameter	The median volumetric diameter of each droplet.
Constant	Use a constant diameter for each droplet.
Rosin-Rammler	Allow each droplet to be sized according to Rosin-Rammler distribution.
Lognormal	Allow each droplet to be sized according to a lognormal distribution.
Rosin-Rammler-Lognormal	Rather than use a constant diameter for each droplet, allow each to be sized according to a combination of Rosin-Rammler and lognormal.
Gamma D	The width of the Rosin-Rammler distribution. The larger the value of gamma, the narrower the droplet size is distributed about the median value.
Sigma D	The width of the lognormal distribution.
Minimum Diameter	Droplets smaller than the minimum diameter are assumed to evaporate in a single time step.
Maximum Diameter	Droplets larger than the maximum diameter are assumed to break up in a single time step.

Injection:

Parameter	Description
Duration	The amount of time a water droplet will remain in the simulation.
Sampling Factor	Sampling factor for the particle output file. A value of -1 uses the FDS default value for this property. Set to an integer greater than 1 to reduce the size of particle output.

Fuel Droplets

Fuel droplets can be used with sprinkler spray models and nozzles to customize the spray. They can also be used in particle clouds and surface types that support particle injection. Evaporating fuel vapor will burn according to the combustion model specified in the active reaction.

To specify a liquid droplet as a fuel droplet, select the species identified by the active reaction as the **Fuel Species**. By default this variable is called **REAC_FUEL**.

Solid Particles

PyroSim provides basic support for specifying solid particles. A solid particle must reference a surface, from which it derives its thermophysical and geometric parameters. A solid particle can be used to model various heat transfer, drag, and vegetation applications. Most of the parameters unique to solid particles must be defined on the Advanced Panel, Chapter 21. For more information, see FDS User Manual (McGrattan, et al., 2013).

Activation

Normally, the insertion of particles into the domain is controlled by the surface or object emitting them, such as by a fan or supply surface or a particle cloud. Alternatively, the insertion of particles can be controlled by a device or other control logic. For more information on controls, see Chapter 155.

Global Parameters

There are two global options relating to particles in the Simulation Parameters dialog. The first option, **Droplets Disappear at Floor**, can be used to prevent droplets from gathering on the floor of the simulation area. The default value for this option is ON. The second option, **Max Particles per Mesh**, can be used to set an upper limit on the number of particles allowed in any simulation mesh.

Particle Clouds

Particle Clouds are box-shaped volumes that can be filled with particles at the start of the FDS simulation or be inserted periodically. To create a particle cloud, on the **Model** menu, click **New Particle Cloud…**. This will show the particle cloud dialog as in Figure 13.1.

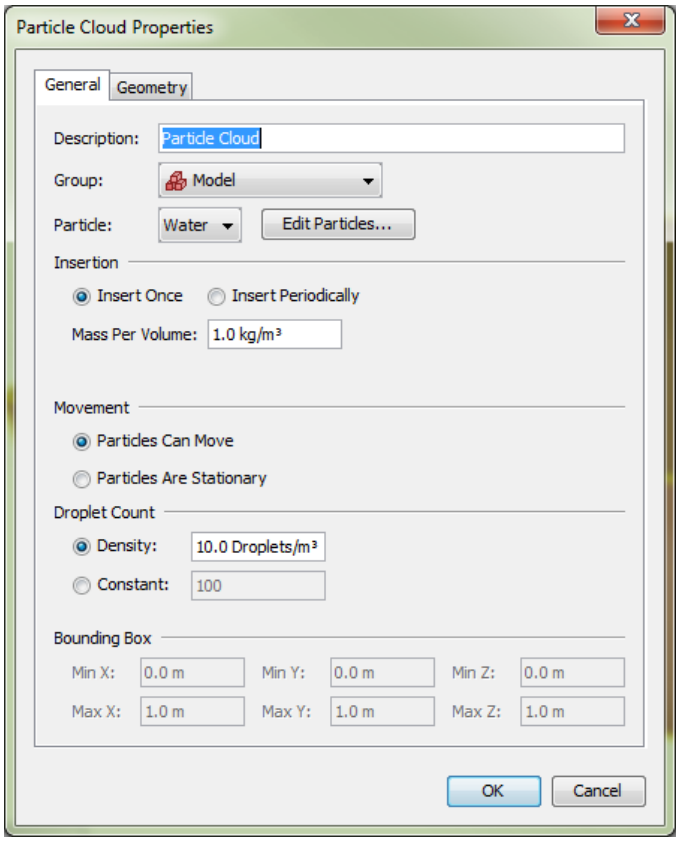


Figure 13.1. Particle Cloud dialog

Particle clouds have the following properties:

- **Particle**: The particle to insert in the volume.
- **Insertion**: How the particles are inserted in the volume.
    - **Insert Once**: Inserts the particles once at the beginning of the simulation.
    - **Insert Periodically**: Inserts particles at an interval throughout the entire simulation.
    - **Mass Per Volume**: The total density of the particles in the volume. This independent of the particle density, which is specified per particle type.
- **Movement**: How the particles move in the volume.
    - **Particles Can Move**: Allows particles to move in the volume.
    - **Particles Are Stationary**: Particles remain stationary and act as obstructions to flow.
- **Droplet Count**: Controls the number of particles in the volume.
    - **Density**: Species the particle count as a density.
    - **Constant**: Species a constant number of particles.

The geometry properties, including the size and location of the volume can be specified on the **Geometry** tab.

Press **OK** to create the new particle cloud. It will appear as a transparent box in the **3D** and **2D Views**.

Chapter 14. Devices

Devices are used to record entities in the model or to represent more complex sensors, such as smoke detectors, sprinklers, and thermocouples. You can make time history plots of device output in PyroSim by opening the **CHID_devc.csv** file.

Devices can be moved, copied, rotated, and scaled using the tools described in Chapter 10 on page 86. Most often, the user will simply select one or more devices, right-click to display the context menu, and click **Copy/Move**. By copying a single device along a line and then copying the line in the normal direction, it is possible to quickly define an array of devices.

After a device is defined, it can be used to activate an object. The value that triggers the activation (setpoint) is defined in the same dialog as the device. This is discussed more in Chapter 155.

Aspiration Detection System

An aspiration detection system groups together a series of soot measurement devices. An aspiration system consists of a sampling pipe network that draws air from a series of locations to a central point where an obscuration measurement is made. To define such a system in FDS, you must provide the sampling locations, sampling flow rates, the transport time from each sampling location, and if an alarm output is desired, the overall obscuration setpoint.

To define the soot measurement devices:

1. On the **Devices** menu, click **New Aspirator Sampler....**
2. Enter the **Name** and **Location** of the sampler.
3. Click **OK** to create the sampler.

To define the aspiration detection system:

1. On the **Devices** menu, click **New Aspirator...**.
2. Give the **Name** and select which **Aspirator Samplers** will be included. For each sampler, provide the data described below.
3. Click **OK** to create the aspirator detector.

Supply the following information for the aspiration detection system, Figure 14.1.

Parameter	Description
Aspirator Name	The name of the aspiration detection system.
Bypass Flow Rate	The flow rate of any air drawn into the system from outside the computational domain.
Transfer Delay	The transport time from the sampling location to the central detector.
Flowrate	The gas flow rate.

Parameter	Description
Location	The coordinates of the aspiration detection system.
Orientation	Not used.
Rotation	Not used.

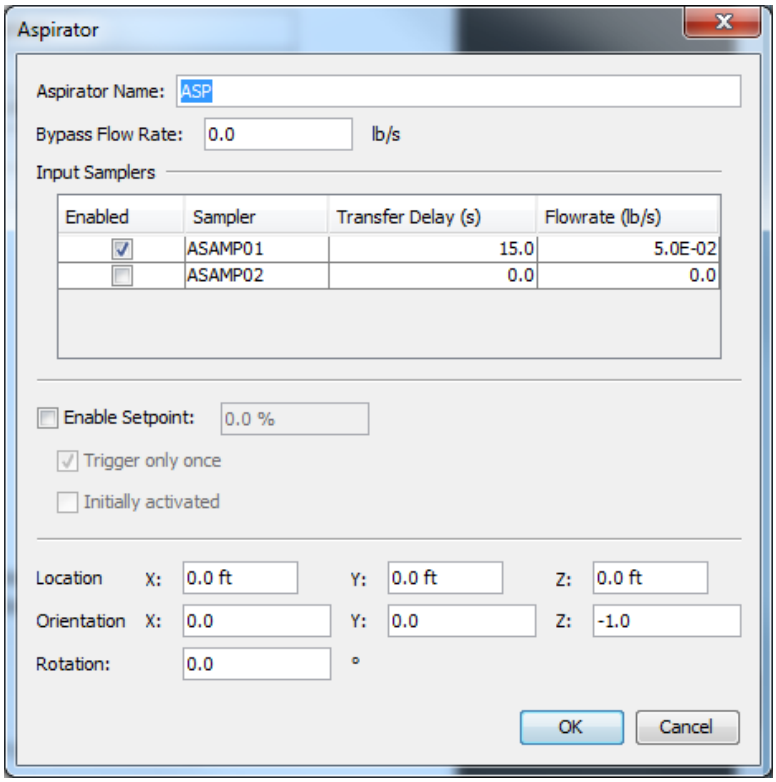


Figure 14.1. Creating an aspirator sampler

The output of the aspiration detection system will be the combined obscuration.

Gas or Solid Phase Device

Simple gas phase and solid phase devices can be used to measure parameters in the gas or solid phase. To define a measurement device:

1. On the **Devices** menu, click **New Gas-phase Device...** or **New Solid-phase Device...** .
2. Enter the **Name** of the device.
3. Select the **Quantity** to be measured.
4. Enter the **Location** of the device. For solid-phase devices, position the device on a surface.
5. Enter the **Normal of Solid** of the device. For solid-phase devices, this is the outward normal to the surface on which the device is attached.
6. The **Rotation** can be left as 0.
7. Click **OK** to create the device.

Thermocouple

To create a thermocouple, on the **Devices** menu, click **New Thermocouple...**.

The thermocouple properties are:

Parameter	Description
Device Name	The name of the thermocouple.
Bead Diameter	The bead diameter of the thermocouple.
Emissivity	The emissivity of the thermocouple.
Bead Density	The material density of the bead. The default value is that for nickel.
Bead Specific Heat	The material specific heat of the bead. The default value is that for nickel.
Location	The coordinates of the device.
Orientation	Not used.
Rotation	Not used.

The output of the thermocouple is the temperature of the thermocouple itself, which is usually close to the gas temperature, but not always, since radiation is included in the calculation of thermocouple temperature.

Flow Measurement

The flow measurement device can be used to measure a flow quantity through an area. To create a flow measuring device, on the **Devices** menu, click **New Flow Measuring Device...**.

The flow measurement device properties are:

Parameter	Description
Device Name	The name of the flow measuring device.
Quantity	The quantity to be measured.
Flow Direction	Select the direction for the measurement as defined by the normal to the measurement plane.
Plane	The axis normal to the measurement plane and the location of that plane on the axis.
Bounds	The coordinates of the area normal to the axis.

The output will be the total flow through the defined area.

Heat Release Rate Device

The heat release rate device measures the heat release rate within a volume. To define a heat release rate device, on the **Devices** menu, click **New Heat Release Rate Device...**.

The heat release rate device properties are:

Parameter	Description
Device Name	The name of the heat release rate device.
Bounds	The coordinates of the volume within which to calculate the heat release rate.

The output will be the total heat release rate within the volume.

Layer Zoning Device

There is often the need to estimate the location of the interface between the hot, smoke-laden upper layer and the cooler lower layer in a burning compartment. Relatively simple fire models, often referred to as two-zone models, compute this quantity directly, along with the average temperature of the upper and lower layers. In a computational fluid dynamics (CFD) model like FDS, there are not two distinct zones, but rather a continuous profile of temperature. FDS uses an algorithm based on integration along a line to estimate the layer height and the average upper and lower layer temperatures. To define a layer zoning device, on the **Devices** menu, click **New Layer Zoning Device...**.

The layer zoning device properties are:

Parameter	Description
Device Name	The name of the layer zoning device.
Measure Layer Height	Enables measurement of the layer height.
Measure Upper Temperature	Enables measurement of the upper layer temperature.
Measure Lower Temperature	Enables measurement of the lower layer temperature.
Path	The coordinates of the end points of a line along which the layer height will be calculated. The two endpoints must lie in the same mesh.

The output will be the quantities selected.

Path Obscuration (Beam Detector) Device

A beam detector measures the total obscuration between points. To define a beam detector device, on the **Devices** menu, click **New Path Obscuration Device...**.

The path obscuration device properties are:

Parameter	Description
Device Name	The name of the path obscuration device.
Path	The coordinates of the end points of a line along which the obscuration will be calculated. The two endpoints must lie in the same mesh.

The output will be the percent obscuration along the path.

Heat Detector

A heat detector measures the temperature at a location using a Response Time Index model. To define a heat detector device, on the **Devices** menu, click **New Heat Detector...**.

The heat detector device properties are:

Parameter	Description
Device Name	The name of the heat detector.
Link	The link defines the activation temperature and the response time index.
Location	The coordinates of the device.
Orientation	Not used.
Rotation	Not used.

The output will be the heat detector temperature.

Smoke Detector

A smoke detector measures obscuration at a point with two characteristic fill-in or “lag” times. To define a smoke detector, on the **Devices** menu, click **New Smoke Detector...**.

The smoke detector device properties are:

Parameter	Description
Detector Name	The name of the smoke detector.
Model	Select the smoke detector type. You can edit the smoke detector parameters to create a new type.
Location	The coordinates of the device.
Orientation	Not used.
Rotation	Not used.

The output will be the percent obscuration per meter.

Sprinkler

Sprinklers can spray water or fuel into the model. To define a sprinkler:

1. On the **Devices** menu, click **New Sprinkler...**. This will display the **Sprinkler** dialog, Figure 14.2.
2. Select the desired options and define required input parameters as described below.
3. Click **OK** to create the sprinkler.

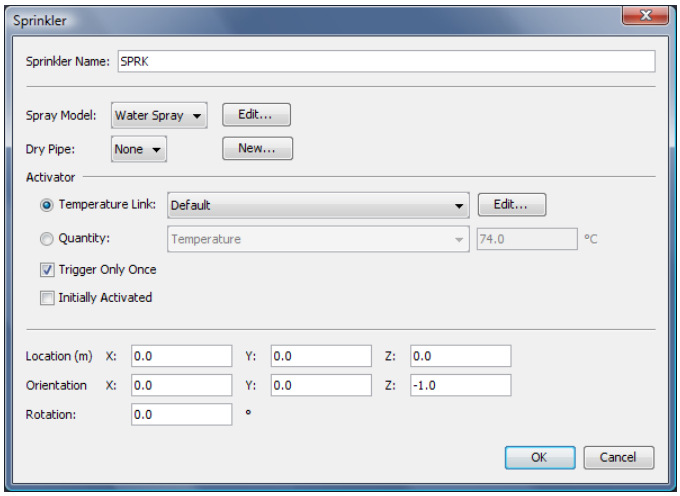


Figure 14.2. Creating a new sprinkler

The sprinkler properties are:

Parameter	Description
Sprinkler Name	The name of the sprinkler.
Spray Model	The spray model defines the particle type (water and fuel are default options), the flow rate, and the jet stream shape.
Dry Pipe	In a dry pipe sprinkler system, the normally dry sprinkler pipes are pressurized with gas. When a link activates in a sprinkler head, the pressure drop allows water to flow into the pipe network. You can create a dry pipe and edit the delay.
Activator	By default the sprinkler is activated by a temperature link, with a response time index. You can edit the activation temperature and the response time index. Alternately, you can select a more general quantity to activate the sprinkler. By default the sprinkler is initially not active and is triggered only once.
Location	The coordinates of the sprinkler.
Orientation	The components of the direction vector.
Rotation	Normally not used for a sprinkler. It could be used to rotate a spray pattern that varies with latitude (circumferentially).

Nozzle

Nozzles are very much like sprinklers, only they do not activate based on the standard RTI model. The can be set to activate by custom control logic.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

Chapter 15. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.
Parameter	**Description**
	velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.
Diameter	The diameter for a circular duct.
Area	The total area for any non-circular duct.
Perimeter	The perimeter of a non-circular duct. Used by the simulator to compute a hydraulic diameter for predicting the flow.

Flow Model options

Parameter	Description
Forward Loss	The friction loss in the duct going from Node 1 to Node 2.
Reverse Loss	The friction loss in the duct going from Node 2 to Node 1.
Roughness	The absolute roughness of the duct material.
Flow Device	The type of flow device you would like to use in the duct.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.
Parameter	**Description**
Damper	A damper device sets an open/closed state for the duct. When a damper control is TRUE it allows normal airflow. When a damper is FALSE, the duct is closed and blocks 100% of the duct area.
Basic Fan	A basic fan represents a fan between the two nodes of the duct. Basic fans are defined on the duct itself, rather than on a fan object, are limited to simpler use cases.
Aircoil	An aircoil represents a heat exchanger between the two nodes of the duct.
Fan	Selecting Fan as the flow device allows you to choose a defined fan object as the airflow device between the nodes of the duct.
Activation	Specifying Activation for either a damper, a fan, or an aircoil attaches a device to set the state of the HVAC component as active or inactive.
Volume Flow	The fixed volumetric flow rate through the duct.
Ramp up time	Used to define a custom time ramp of the duct flow rising to the Volume Flow.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.
Parameter	**Description**
Fan	Selects a defined HVAC Fan object to use as the flow device.
Flow Direction	Choose the direction of airflow. By default, air moves from Node 1 to Node 2.
Aircoil	Selects a defined HVAC Aircoil object to use as the flow device.

HVAC Node

An HVAC Node represents a either a joining of two or more HVAC Ducts, or the meeting point between a duct and the PyroSim computational model. To create an HVAC Node:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `NODE`.
4. Click **OK**.

You can now edit the node.

Parameter	Description
Filter	Select the HVAC Filter to be used at the node.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.
Parameter	**Description**
Auto	Selecting Node Type Auto tells PyroSim to predict the node type based on its interaction with other HVAC objects.
Internal	An Internal node is one connected to only ducts. An internal node must be connected to at least two ducts.
Ambient Endpoint	A node specified as an Ambient Endpoint is connected to at least one duct and is equivalent to an OPEN surface.
Vent Endpoint	If you select a node as a Vent Endpoint, you can select a vent with surface type HVAC, and the node will be the meeting point between the HVAC system and the rest of the PyroSim model.
Location	The 3D coordinates of the node. If the location is not specified, the default values of 0, 0, 0 will be used by the simulator.
In Loss	The flow loss for gases entering the HVAC system.
Out Loss	The flow loss for gases exiting the HVAC system.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New...**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

HVAC Fan

An HVAC Fan is used to generate airflow in a HVAC network. A fan is specified between two nodes by selecting it as the Flow Device for an HVAC Duct. Note that an HVAC Fan is a class of object, and a single fan definition can be used by any number of ducts.

To create an HVAC Fan:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New...**.
3. In the **Type** box, select `FAN`.
4. Click **OK**.

Parameter	Description
Activation	Attach a control logic device to the fan.
Flow Loss	The loss through the fan when the fan is not operational.
Initial Ramp up time	Specify either a Tanh or a t^2 ramp up time to the flow rate.
Maximum Flow Rate	The maximum volumetric flow rate used for the quadratic fan model.
Maximum Pressure	The maximum stall pressure of the fan used for the quadratic

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New...**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.
Parameter	**Description**
	fan model.
Volume Flow Rate	The fixed volumetric flow of the fan.
Fan Curve	Used to specify a table of pressure drops across the fan versus the volumetric flow rates.

HVAC Filter

An HVAC Filter is used to stop gaseous species from circulating in the HVAC system. A given filter can limit the flow of any number of valid species defined in the model. Note that an HVAC Filter is a class of object, and a single filter definition can be used by any number of nodes.

To create an HVAC Filter:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New...**.
3. In the **Type** box, select `FAN`.
4. Click **OK**.

Parameter	Description

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.
Parameter	**Description**
Clean Loss	The flow loss across the filter at zero loading.
Loss	The loss as function of the clean loss and the species loadings/multipliers for the filter.
Loss (Custom)	Specify a ramp loss table as a function of total loading.
Species	The tracked species to be filtered.
Efficiency	The efficiency at which the species is filtered.
Initial Loading	The loading on the filter for the given species at t = 0.
Loading Multiplier	A factor used for calculating the overall flow loss at the filter.

HVAC Aircoil

An HVAC Aircoil is a device that provides a heating or cooling element to an HVAC system. An aircoil is specified between two nodes by selecting it as the Flow Device for an HVAC Duct. Note that an HVAC Aircoil is a class of object, and a single aircoil definition can be used by any number of ducts.

To create an HVAC Aircoil:

1. On the **Model** menu, click **Edit HVAC...**.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

2. Click **New…**.
3. In the **Type** box, select `AIRCOIL`.
4. Click **OK**.

Parameter	Description
Activation	Attach a control logic device to the aircoil.
Heat Exchange Rate	A constant exchange rate between the aircoil and the air across it. A negative value represents heat removal.
Ramp-Up Time	Specify either a Tanh or a t^2 ramp up time to the heat exchange rate.
Coolant Specific Heat	The specific heat of the working fluid.
Coolant Mass Flow Rate	The mass flow rate of the working fluid.
Coolant Temperature	The inlet temperature of the working fluid.
Heat Exchanger Efficiency	A value from 0 to 1 representing the efficiency of the aircoil. A value of 1 indicates the exit temperatures on both sides of the heat exchanger will be equal.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

HVAC Vents

HVAC Vents are used to represent the junction between the HVAC system and the rest of the computational model. See Chapter 7 for more information on using vents. To define an HVAC Vent:

1. On the **Model** menu, click **New Vent...**.
2. In the **Surface** box, select **HVAC**.
3. Click the **Geometry** tab. Specify the appropriate 2D geometry of the vent.
4. Click the **HVAC Properties** tab. If you want the flow from the vent to go in a direction not perpendicular to the vent, click **Louver**. Specify the values **X**, **Y**, and **Z** as a vector of the direction you would like the flow to be directed.
5. Click **OK**.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

Control Logic

Objects can be set to activate or deactivate during the simulation using activation events. Activation events are the control logic system in FDS and can be set on each geometric simulation object (e.g. walls, holes, vents) using the **Activation** option in the object's **Properties** dialog. PyroSim supports activation events based on time and input devices. Some uses of activation events include:

- Causing a door obstruction to be removed (i.e. opened) from the simulation at a particular time,
- Causing a window obstruction to be removed (i.e. break) when a heat detector triggers, and
- Causing a ventilation system to activate when any of several smoke detectors activates

To open the **Activation Controls** dialog (shown in Figure 0.1): On the **Devices** menu, click **Edit Activation Controls...**.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New...**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

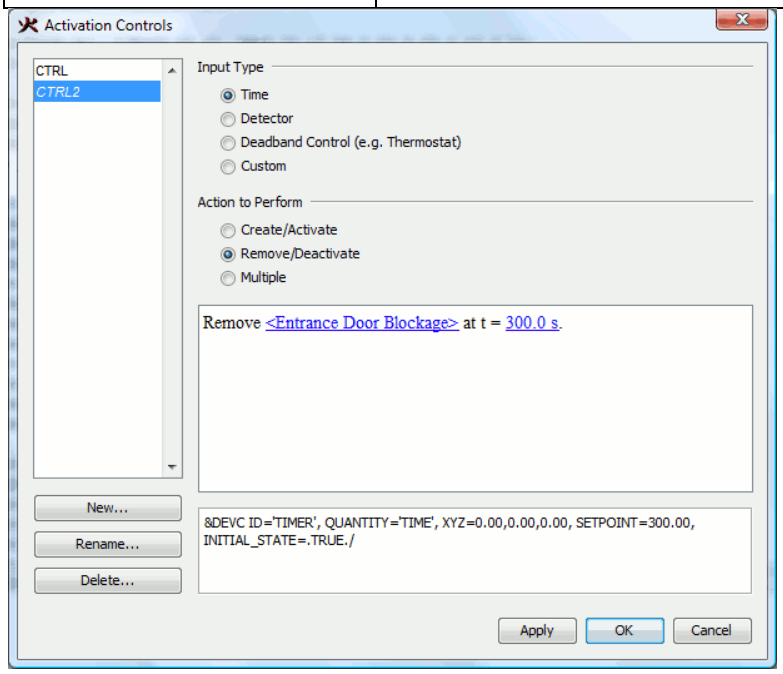


Figure 0.1. The Activation Controls dialog

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

Creating Activation Controls

Creating controls in PyroSim takes 3 steps:

1. Select an input type (time, detector, deadband, or custom). This is the source of the signal that will trigger the control.
2. Choose an action to perform (e.g. create an object).
3. Set specific inputs for the control based on a pattern created by steps 1 and 2.

After selecting an input type and an action, a pattern (in sentence form) for describing the control logic will appear in the dialog. Some key words and numbers will be drawn in blue and underlined. Any blue text can be clicked to modify the behavior of the specific control.

Figure 0.2 shows the selector popup for objects. Objects are selected by name. To quickly find objects in the selector popup, you can type the first few letters of the object's name.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

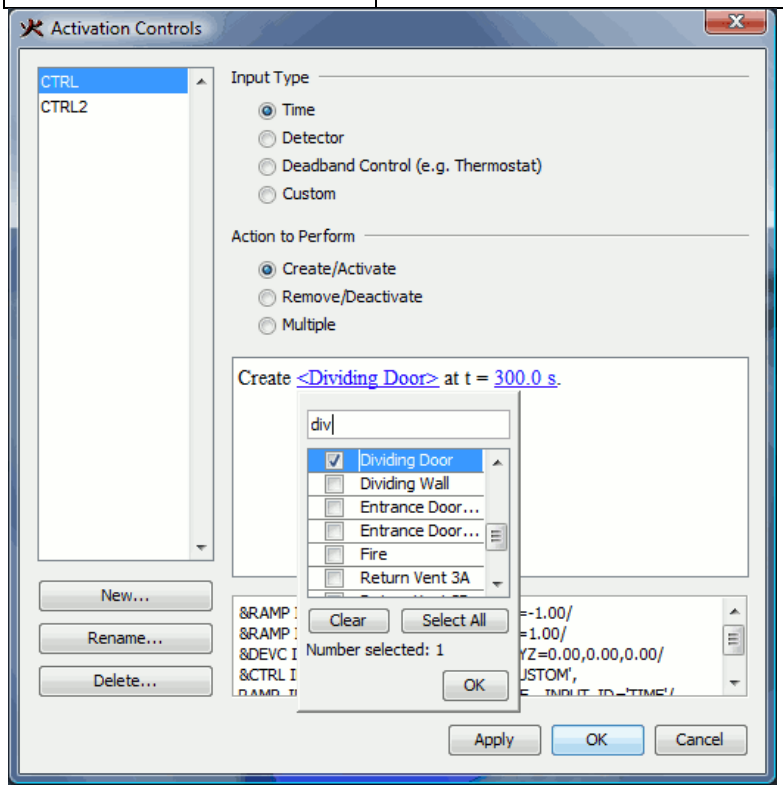
A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New...**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.



Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

Figure 0.2. The object popup in the Activation Controls dialog

Activation controls are stored separately from specific geometric objects. This makes it possible to bind an object to a control after it has been created. You can use the **Activation** box in an object's properties editor to bind that object to an existing activation control, or even create a new control directly. Figure 0.3 shows the activation control in the object properties dialog for a hole.

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New...**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

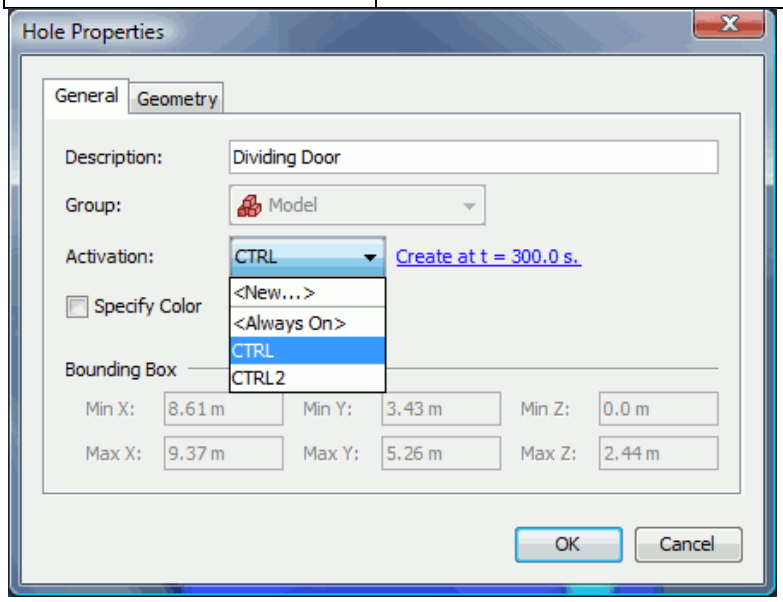


Figure 0.3. The Activation Controls dialog

Once a control has been bound to an object (or objects) any objects linked to that control will show a text description of the control in their properties editor. This text will be shown in blue and underlined

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

and can be clicked to edit the activation control. Changes made to the activation control will impact all referencing objects.

Time-based Input

To create or remove an object at a specific time, select **Time** for the **Input Type** in the **Activation Controls** dialog. When using time as the input, objects can be created at a specific time, removed at a specific time, or be created and removed periodically throughout the simulation. To create or remove the objects once, select **Create/Activate** or **Remove/Deactivate** under the **Action to Perform**. To create/remove the object periodically, select **Multiple** under the **Action to Perform**. When performing multiple timed events, the creation and removal and times at which they occur are specified in the table at the bottom of the dialog. The create and remove events should alternate as time increases.

Detector-based Input

To create or remove some objects based on a device in the model, the device must first have a setpoint enabled. To specify a setpoint, perform the following:

1. Create a new device or open the properties for an existing device.
2. Check the box next to **Enable Setpoint:**
3. Enter the desired value at which the device will trip.
4. Choose the options for the detector:

Chapter 1. HVAC Systems

In PyroSim, HVAC (Heating, Ventilation, and Air Conditioning) systems are specified as networks of ducts, nodes, and some combination of fans, aircoils, and filters.

HVAC Duct

A duct is required for any HVAC system. At a basic level, a duct represents the joining of two HVAC Nodes. To define an HVAC Duct:

1. On the **Model** menu, click **Edit HVAC...**.
2. Click **New…**.
3. In the **Type** box, select `DUCT`.
4. Click **OK**.

You can now edit the duct:

Properties

Parameter	Description
Node	The names of nodes on either side of the duct. A positive velocity is described as flowing from Node 1 to Node 2.
Fixed	By default, PyroSim estimates the length of the duct based on the straight line geometry between the specified nodes. If for any reason this is not a sufficient representation of the model, you may explicitly specify a fixed length for the duct.

- **Trigger only once** - the detector will only trigger once at the setpoint during the entire simulation.
- **Initially activated** - if this is unchecked, the detector is untriggered at the start of the simulation and will turn on when the setpoint is reached for the first time. If this is checked, the detector is triggered at the start and will turn off when the setpoint is reached.

5. Click **OK** to close the device dialog.

Once the desired devices have been given a setpoint, they can be used as inputs to the control logic expression. Now in the **Activation Controls** dialog, select **Detector** as the **Input Type**. The detector can be used to **Create/Activate** or **Remove/Deactivate** the desired objects when the detector either activates or deactivates. If more than one detector is to be used to activate the objects, the descriptive sentence can be used to decide if the objects should trigger when any, all, or a certain number of the devices activate.

Chapter 16. Output Controls

In this chapter we describe the simulation output options available in PyroSim. Each of these options is located in the **Output** menu.

Solid Profiles

Solid profiles measure quantities (e.g. temperature, density) as they extend into solid objects. The output file for this measurement device will be named `CHID_prof_n` where `CHID` is the job ID and `n` is the index of the solid profile. This output file contains the data necessary to create an animated 2D chart of the quantity as it extends into the object over time. PyroSim does not currently support displaying this output file.

To generate solid profile output, on the **Output** menu, click **Solid Profiles...**. Each solid profile requires the following parameters:

Parameter	Description
ID	The name of this solid profile entry.
X, Y, Z	The coordinates of a point on the face that will be examined by this solid profile.
ORIENT	The direction of the face that will be examined by this solid profile. To generate solid profile output for the top of an object, this value will be Z+. This parameter prevents any ambiguity that might result from a point that lies on two adjoining faces.
QUANTITY	The quantity that will be measured in this solid profile.

Note

The surface to be measured must be heat-conducting. If the surface on the specified face is not heat-conducting, FDS will issue an error and exit before running the simulation.

Slices

Slices or *slice planes* measure gas-phase data (e.g. pressure, velocity, temperature) on an axis-aligned plane. This data can then be animated and displayed using Smokeview (Figure 16.1).

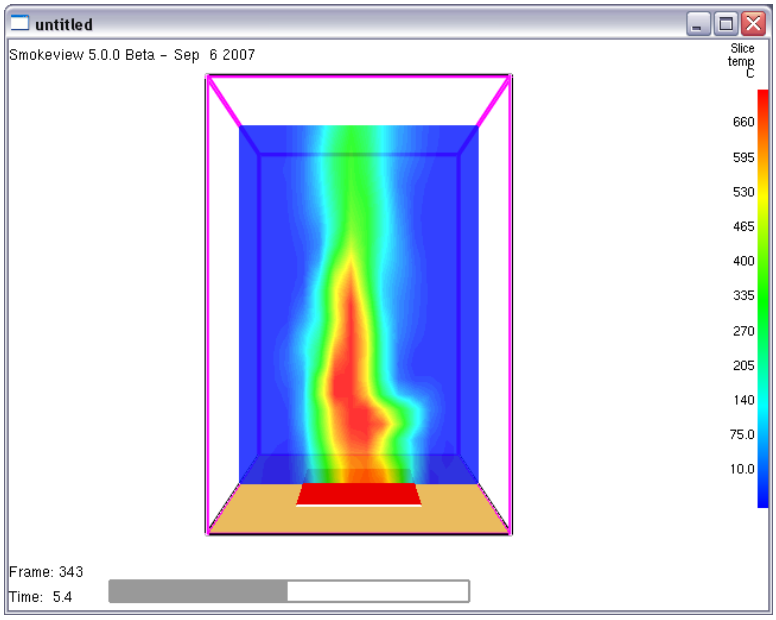


Figure 16.1. An example of a slice plane shown in Smokeview.

To generate animated slice planes, on the **Output** menu, click **Slices...**. Each slice plane requires the following parameters:

Parameter	Description
XYZ Plane	The axis (X, Y, or Z) along which to place the slice plane.
Plane Value	The value along the specified axis where the plane will be placed.
Gas Phase Quantity	The quantity that this plane will measure. This list includes built-in options such as temperature as well as dynamic options such as those base on particles.
Use Vector?	Setting this option to YES will cause FDS to generate additional flow vector data for this slice.

Slice files may be viewed in Smokeview by selecting **Load/Unload->Slice** file. To view the vector representation, select **Load/Unload->Vector** slices.

Boundary Quantities

Boundary quantities provide a way to visualize output quantities (e.g. temperature) on the walls of every obstruction in the simulation. This data can be animated and visualized in Smokeview (Figure 16.2). Since the data applies to all surfaces in the simulation, no geometric data needs to be specified.

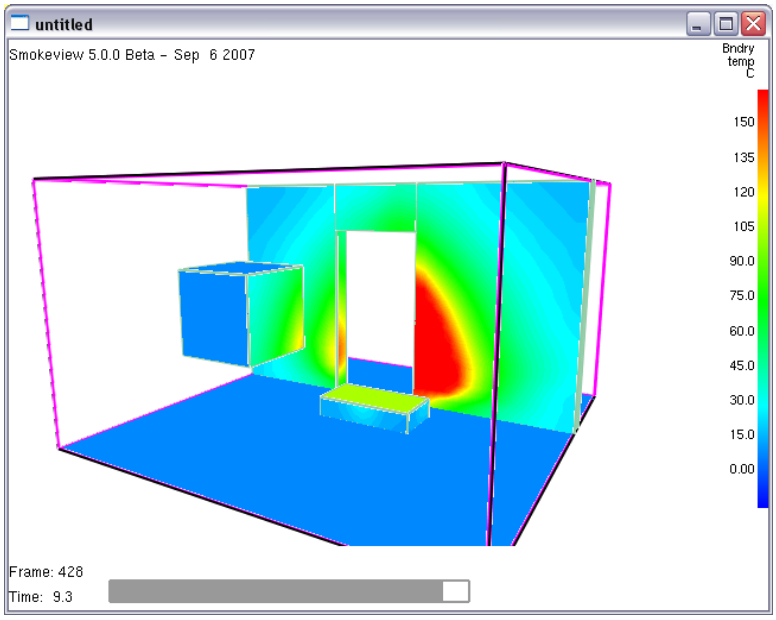


Figure 16.2. An example of a boundary quantity shown in Smokeview

To generate boundary quantity data, on the **Output** menu, click **Boundary Quantities...**. In the **Animated Boundary Quantities** dialog, you can select each quantity you would like to be available for visualization.

To view boundary data in Smokeview (e.g. wall temperature), right-click to open the menu, then select: **Load/Unload->Boundary File->WALL_TEMPERATURE**.

Isosurfaces

Isosurfaces are used to plot the three dimensional contour of gas phase quantities. This data can be animated and visualized in Smokeview (Figure 16.3).

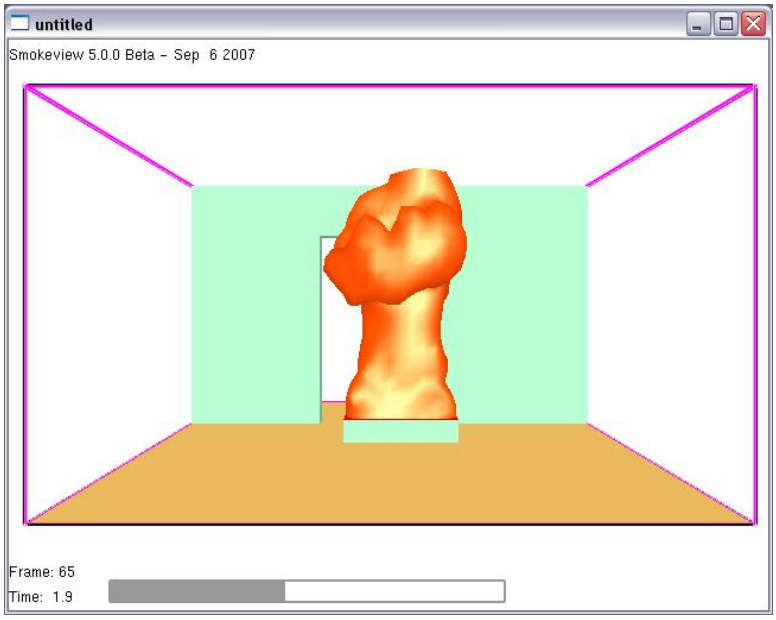


Figure 16.3. An example of an isosurface shown in Smokeview

To generate isosurface data, on the **Output** menu, click **Isosurfaces...**. In the **Animated Isosurfaces** dialog, you can select each quantity you would like to be available for visualization. Then you must enter values at which to display that quantity in the **Contour Values** column. If you enter more than one contour value, each value must be separated by the semi-colon character (**;**). Once you have finished typing the value, press enter.

To view isosurface data in Smokeview, right-click to open the menu, then select: **Load/Unload->Isosurface File->TEMPERATURE**.

Plot3D Data

Plot3D is standard file format and can be used to display 2D contours, vector plots, and isosurfaces in Smokeview (Figure 16.4).

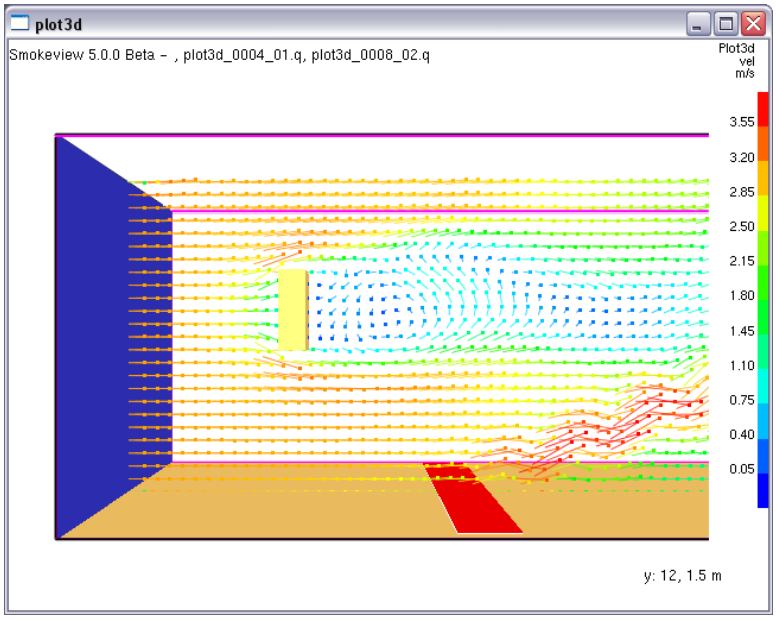


Figure 16.4. A velocity Plot3D data shown in Smokeview

By default, Plot3D data will be generated for the following quantities: Heat Release Rate per Unit Volume (HRRPUV), Temperature, U-Velocity, V-Velocity, and W-Velocity. To change the Plot3D output quantities, on the **Output** menu, click **Plot3D Data...**. In the **Plot 3D Static Data Dumps** dialog, select the output quantities to generate. Since FDS supports a maximum of five Plot3D output quantities, you must remove some existing quantities to add new ones.

Statistics

Statistics output is an extension of the devices system. You can insert a statistics gathering device and it will output data about the minimum, maximum, and average value of a particular quantity in one or more mesh. This data can then be viewed in a 2D chart using PyroSim (Figure 16.5).

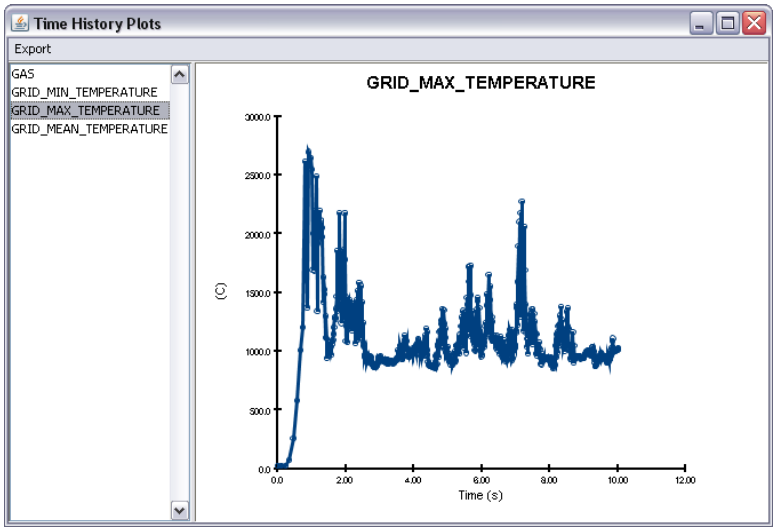


Figure 16.5. An example of statistic output

To generate statistics data for some region, on the **Output** menu, click **Statistics...**, then click **New...**. Once a quantity is selected, some combination of the following options is available depending on whether the quantity is gas or solid-phase and what units are output by the quantity:

Parameter	Description
Quantity	The quantity that will be measured. This value is set when you create the statistics entry and cannot be modified.
Mean	Select this option to output the average value of the measured quantity over time in a mesh.
Minimum	Select this option to output the minimum value of the measured quantity over time in a mesh.
Maximum	Select this option to output the maximum value of the measured quantity over time in a mesh.
Volume Mean	This option is available for gas-phase quantities only. It is similar to “Mean,” but each cell value is weighted according to its relative size.
Mass Mean	This option is available for gas-phase quantities only. It is similar to “Mean,” but each cell value is weighted according to its relative mass.
Volume Integral	This option is available for gas-phase quantities whose units involve m^{-3}, such as heat release rate per unit volume.
Area Integral	This option is available for gas-phase quantities whose units involve m^{-2}. If this option is selected, a recording area rather than a recording volume must be defined.
Surface Integral	This option is available for solid-phase quantities whose units involve m^{-2}, such as heat and mass fluxes.
Recording Volume	This is available for gas-phase quantities as long as the **Area**

Parameter	Description
	Integral option is not selected. This defines a volume over which the statistics will be taken.
Recording Area	This is available for gas-phase quantities if the **Area Integral** option is selected. This defines an area over which the statistics will be taken.
Recording Mesh	This is available for solid-phase quantities. Select the mesh for which you would like to output this statistical data. A proper location will automatically be chosen for the device in the FDS output file.

The output file for measurement devices will be named `CHID_devc.csv` where `CHID` is the job ID.

Note

When using statistics data, it is important to consider nuances of FDS's numerical solver. For instance, the minimum statistic is sensitive to numerical errors in the solver during species transport and will sometimes report artificially low values.

Chapter 17. Evac

FDS contains an evacuation model that makes it possible to perform a coupled fire and evacuation simulation. Documentation relating specifically to FDS+EVAC is not included with PyroSim. To learn more, please visit the FDS+EVAC web site at: http://www.vtt.fi/proj/fdsevac/index.jsp

PyroSim supports FDS+EVAC by optionally activating user interface controls in the MESH editor as well as the geometric object (obstructions, holes, vents) editors. Also, for each FDS+EVAC namelist, PyroSim provides a manager dialog in the **Evac** menu.

When FDS+EVAC support is enabled, the **Evac** menu options will be enabled, the FDS+EVAC UI components will be present in the MESH and object editors, and FDS+EVAC options will be included with any FDS input files generated by PyroSim - as seen in the record view, when exporting FDS input files, and when running simulations. When FDS+EVAC support is disabled, the **Evac** menu options will be disabled, FDS+EVAC UI components will not be present in the MESH and object editors, and FDS+EVAC namelists will be excluded from any FDS input files generated by PyroSim. FDS+EVAC options within MESH, OBST, HOLE, and VENT records will continue to be written to prevent modification of the fire model. Disabling FDS+EVAC within PyroSim does not cause FDS+EVAC data to be lost, this data will remain unchanged until FDS+EVAC is enabled again.

Using FDS+EVAC

By default, options relating to FDS+EVAC are disabled in PyroSim. If you load a model that uses FDS+EVAC features or if you import an FDS file that contains FDS+EVAC records, PyroSim will automatically activate the FDS+EVAC features. To manually activate FDS+EVAC support in PyroSim:

- On the **Evac** menu, click **Enable FDS+EVAC**

FDS+EVAC works by establishing flow fields using special 2D evacuation-specific meshes and low-powered intake vents at the building exits. To run the evacuation model with a fire model you must:

- Define new meshes that will be used specifically for the evacuation simulation. For these meshes, the **Evacuation** box must be checked.
- Define an exhaust (outflow) surface used to create the evacuation flow field. The FDS+EVAC manual recommends this surface have a velocity of `1.0e-6 m/s` and a ramp time of `0.1 s` using a Tanh curve.
- Place vents at the exit and door locations and assign to them the outflow surface. Using the **Evac** tab in the vent editor, specify that the vents will be used in the evac simulation only. This will prevent the vents from influencing the fire model. Because vents must be placed against solid objects, it may be necessary to back these vents with an evac-only obstruction.
- Create the DOOR and EXIT objects using the editors available on the **Evac** menu. Usually these will be placed in the same location as the evacuation exhaust vents.
- Add occupants to the simulation using the **Initial Positions** dialog available on the **Evac** menu.

To take advantage of the EVAC (Initial Positions) feature that limits known exits, it is necessary to create an evacuation mesh for each exit. This secondary mesh must be attached to the alternate exit vent, allowing it to receive an alternate flow field. This field will then be used by occupants that select the alternate exit.

Where to Find FDS+EVAC Records in PyroSim

PyroSim supports all the FDS+EVAC namelists and parameters given in the FDS+EVAC Users Guide (FDS v5.3.1, EVAC v2.1.2). This section shows how the PyroSim dialogs relate to the FDS input records.

FDS+EVAC uses several inputs that should only defined once in the input file. This global data includes FDS+EVAC entries on the TIME and MISC namelist groups as well as global data that is defined using a PERS namelist. This data can be edited in PyroSim using the **Evac Properties** dialog.

The EVACUATION, EVAC_HUMANS, and MESH_ID records associated with MESH, OBST, HOLE, and VENT records can be edited using the usual editor for that record. The controls used to edit these parameters will not be available when the FDS+EVAC mode is disabled.

The FDS+EVAC-specific namelists can be edited by selecting the corresponding option on the **Evac** menu. The mapping for these dialogs is shown below.

PyroSim Dialog	FDS+EVAC Namelist
Person Types	PERS
Initial Positions	EVAC
Evac Holes	EVHO
Exits	EXIT
Entrances	ENTR
Doors	DOOR
Corridors	CORR
Incline/Stairs	EVSS

When viewing the descriptive text for a user interface element in PyroSim, you can learn the specific FDS+EVAC parameter by moving the mouse pointer over the text. The popup text will show the FDS+EVAC record name.

For detailed information on all of the FDS+EVAC parameters, please refer to the FDS+EVAC users guide.

Chapter 18. Running the Simulation

All aspects of running an FDS simulation can be performed through the PyroSim user interface, including setting up simulation parameters, executing single- and multi-threaded simulations, running a remote cluster simulation, and resuming previously stopped simulations.

Simulation Parameters

Before running a simulation, FDS simulation parameters should be adjusted to fit the problem. This can include parameters such as simulation time, output quantities, environmental parameters, conversion of angled geometry to blocks, and miscellaneous simulator values.

To edit the simulation parameters, on the **FDS** menu, select **Simulation Parameters…**. This shows the simulation parameters dialog. The parameters are split into several categories, with each category on another tab of the dialog.

Time

All time-related values can be entered on the **Time** tab as shown in Figure 18.1.

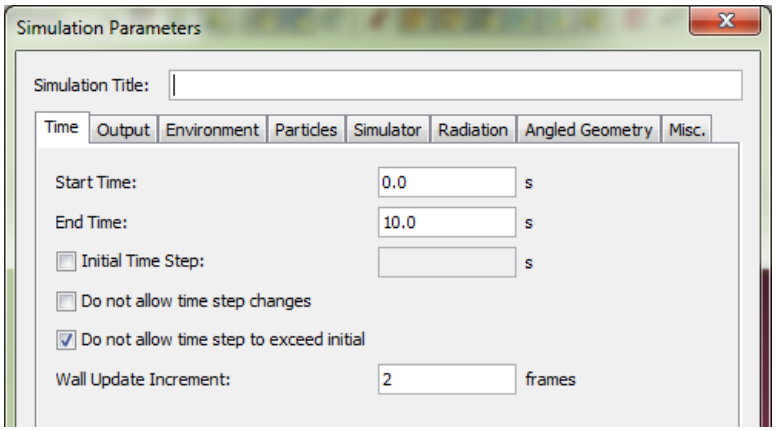


Figure 18.1. Time tab of the simulation parameters dialog

- **Start Time:** a remapping of simulation time, t=0, to a different time. This is used to format the output time, and can be useful for recreation scenarios.
- **End Time:** the ending simulation time.
- **Initial Time Step:** Overrides the default time step.
- **Do not allow time step changes:** Prevents FDS from dynamically altering the time step.
- **Do not allow time step to exceed initial:** Prevents FDS from allowing the time step to go above the initial time step.

Output

The **Output** tab provides fine-grained control of how output values are recorded.

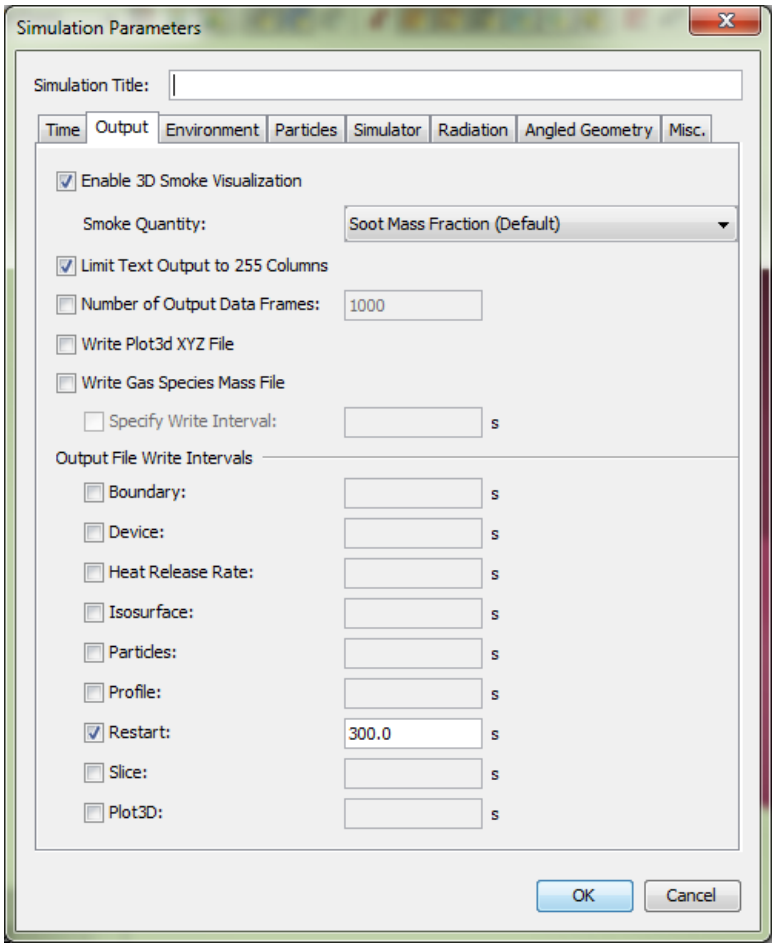


Figure 18.2. Output tab of the simulation parameters dialog

- **Enable 3D Smoke Visualization:** whether to show smoke in the results. If enabled, the visualization can be based on various species in the model.
- **Limit Text Output to 255 Columns:** Limits how many columns are written to CSV output files.
- **Output File Write Intervals:** Specifies intervals at which to write to various output files.

Environment

The **Environment** tab enables various ambient environmental properties to be set as shown in Figure 18.3

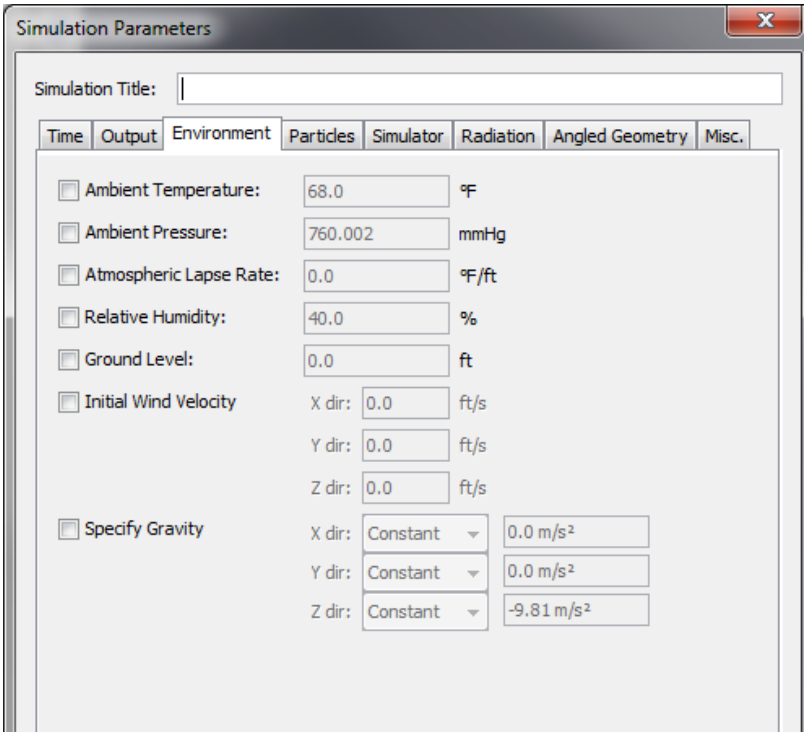


Figure 18.3. Environment tab of the simulation parameters dialog

A unique aspect of this tab is the specification feature for gravity. Gravity, in each of the X, Y, and Z directions, can be defined as a ramped function. This allows users to model complex behavior of gravity in tunnel or space applications where spatial or temporal variations in direction may change the magnitude vector. Each ramp can be set to vary as a function of either the position along the X direction, or time.

While this tab provides control over default environmental conditions, different temperatures, pressures, and mass fractions of species can be specified in various sub-regions of the model by using **Init Regions**.

To create an init region, exit the Simulation Parameters dialog, and on the **Model** menu, choose **New Init Region…**. This opens the Initial Region dialog as shown in Figure 18.4. Specify the desired temperature, pressure, or mass fraction of species to override in the region on the **General** tab and enter the volume parameters on the **Geometry** tab. Press **OK** to create the init region.

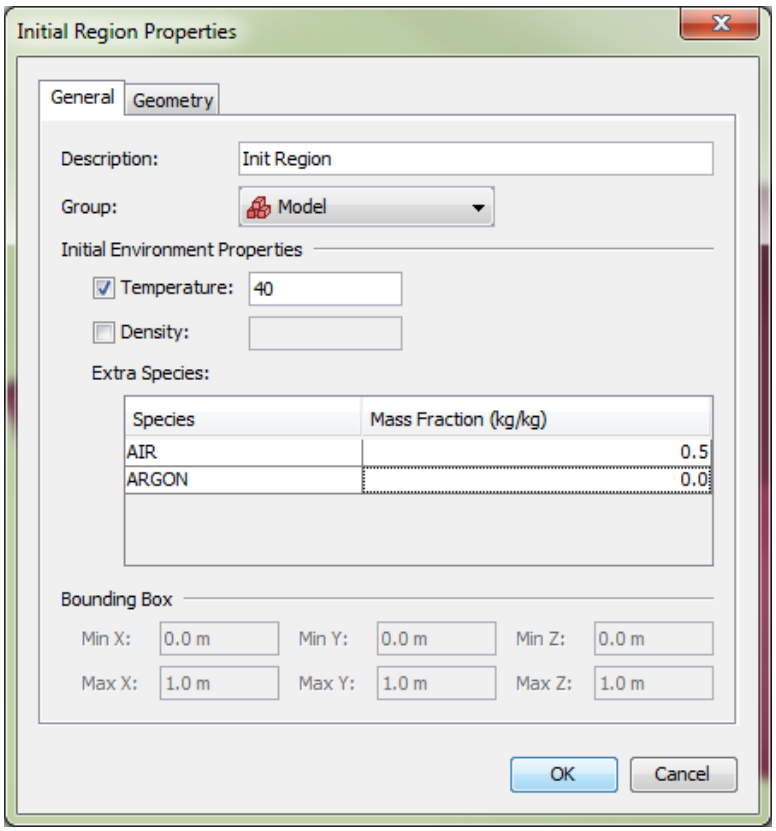


Figure 18.4. Init Region dialog

Simulator

The **Simulator** tab provides control over the simulator used in FDS. Refer to the FDS User Manual (McGrattan, et al., 2013) for more information on various parameters.

Radiation

The **Radiation** tab provides control over radiation parameters used in FDS. Refer to the FDS User Manual (McGrattan, et al., 2013) for more information on radiation parameters.

Angled Geometry

PyroSim allows obstructions and holes to be drawn that are not aligned with the solution mesh needed by FDS. To write the FDS input file, PyroSim must convert these objects to axis-aligned blocks, first. PyroSim will either do this automatically when the FDS input file is generated, or this can be done manually for individual objects by right-clicking the object and selecting **Convert to Blocks…**.

The A**ngled Geometry** tab of the simulation parameters dialog provides default parameters that control conversion of obstructions and holes into blocks for the FDS input file as shown in Figure 18.5.[1]

[1] If objects are converted manually through the right-click menu, a dialog with the same conversion parameters is shown to the user that is initialized with the values from the angled geometry tab of the simulation parameters dialog.

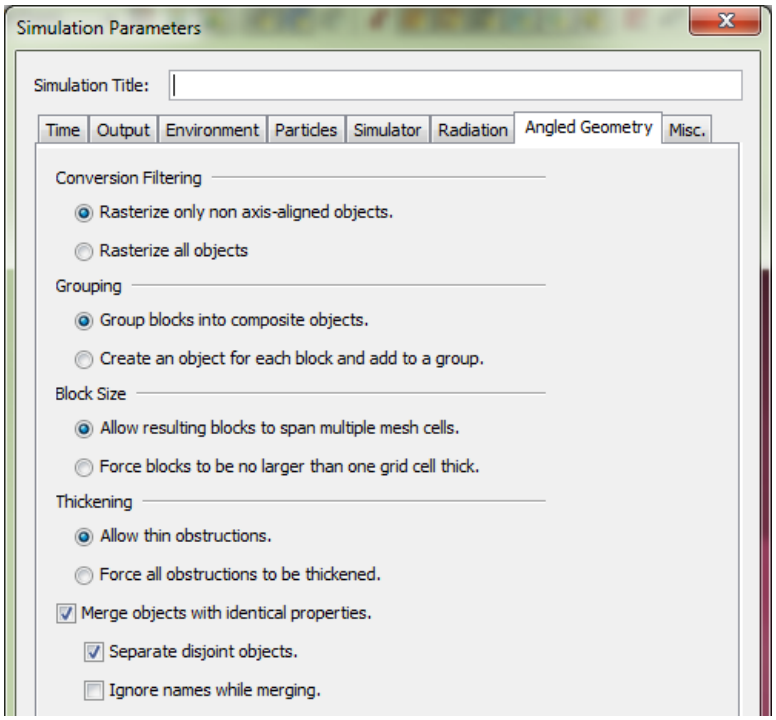


Figure 18.5. Angled Geometry tab of the simulation parameters dialog

- **Conversion Filtering:** Controls which objects are converted into blocks.
  - **Rasterize only non axis-aligned objects [default]:** This prevents objects that are already axis-aligned blocks from being processed in the conversion engine.[1]
  - **Rasterize all objects:** Forces all obstructions and holes, regardless of shape, to be converted to blocks.
- **Grouping:** Controls how resulting objects are created after being converted to blocks. This is more relevant to manual conversion of objects to blocks.
  - **Group blocks into composite objects [default]:** For each converted object (such as a wall), creates one resulting object that is a composite of all the sub-blocks.
  - **Create an object for each block and add to a group:** Creates a new PyroSim object for each resulting block. These objects are then added to a group representing the original object.
- **Block Size:** Controls how large resulting blocks may be. See Figure 18.6.
  - **Allow resulting blocks to span multiple mesh cells [default]:** Resulting adjacent blocks with the same properties are merged into one block, vastly reducing the number of produced blocks.
  - **Force blocks to be on larger than one grid cell thick:** Blocks will not be merged. This may create great numbers of blocks that will take additional memory but have the advantage of being more easily deleted.

[1] Using this option skips axis-aligned blocks, but it does not check that the boundaries of the skipped objects are aligned with mesh cell boundaries.

- **Thickening:** Controls whether objects are allowed to be thin.
  - **Allow thin obstructions [default]:** Allows objects to become thin as shown in Figure 18.7. This may be overridden for obstructions by turning on **Thicken** in the obstruction properties dialog.[1]
  - **Force all obstructions to be thickened:** Prevents all obstructions from becoming thin.
- **Merge objects with identical properties [default=true]:** Allows blocks to be merged across source objects if their resulting blocks have similar properties. For instance, if this is true and two walls are drawn with similar properties, their converted blocks will be merged into the same group or composite object. This can further reduce the number of resulting objects.
  - **Separate disjoint objects [default=true]:** Prevents objects' block from being merged if their blocks do not touch.
  - **Ignore names while merging [default=false]:** Controls whether names are considered when determining whether objects are "similar."

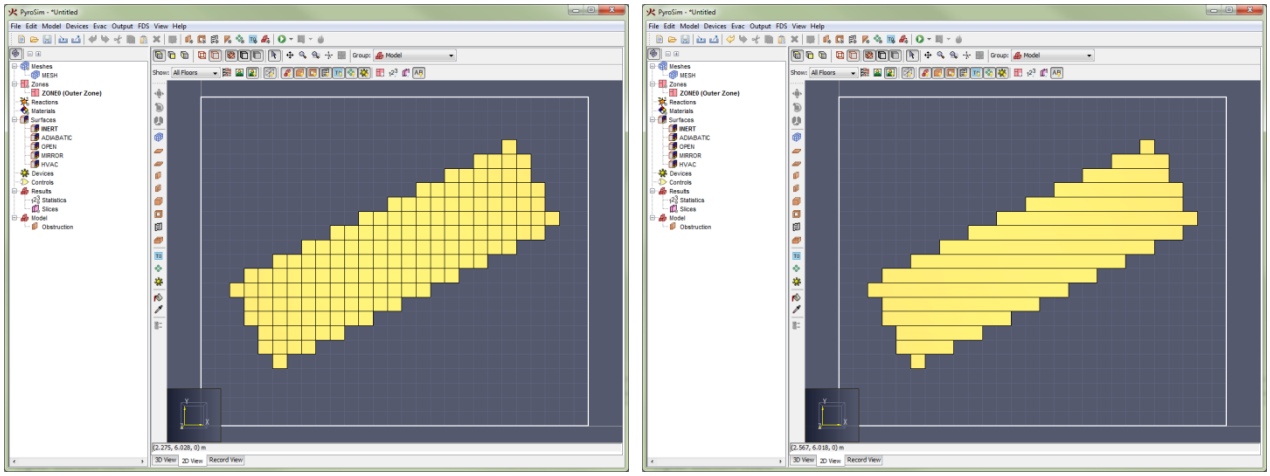


Figure 18.6. Merging converted blocks

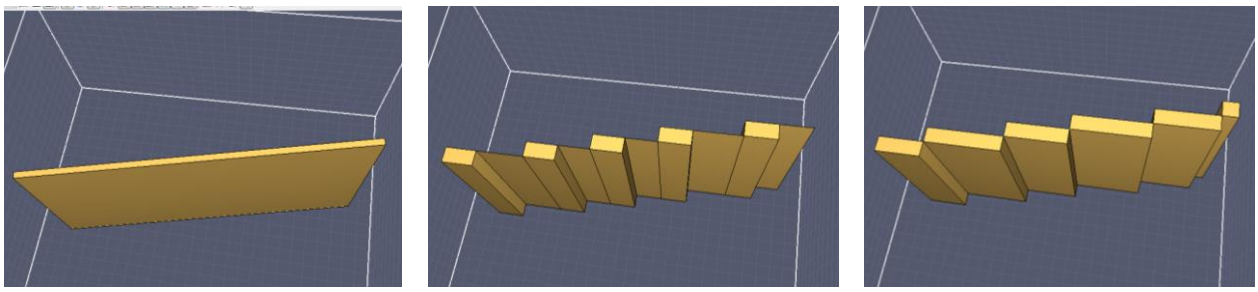


Figure 18.7. Effect of thickening on converted blocks

Misc Tab

This tab allows some miscellaneous simulation and model properties to be set.

[1] FDS does not allow thin obstructions to have vents attached to them.

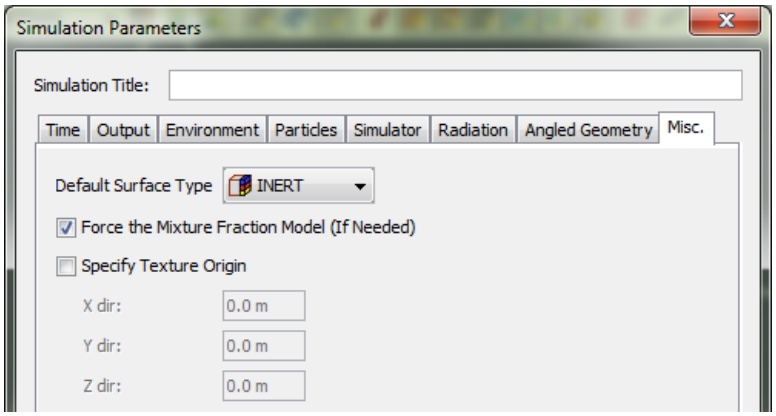


Figure 18.8. Misc. tab on the simulation parameters dialog

- **Default Surface Type:** This specifies the surface to apply to mesh boundaries.
- **Force the Mixture Fraction Model (If Needed):** This detects if an output quantity is being used, such as by a device,

Single-thread Execution

Once you have created a fire model, you can run the simulation from within PyroSim. FDS actions can be accessed from either the **FDS** menu or the main toolbar, as shown in Figure 18.9. To begin a single-processor simulation, on the **FDS** menu, click **Run FDS...** or click from the main toolbar. NOTE: By default, PyroSim will not automatically save the current model to disk unless indicated to do so under **File->Preferences...** under the **FDS** tab.

Figure 18.9. The FDS Toolbar

PyroSim will create a sub-directory of the current PyroSim file to store FDS input and results. So for instance, if the PyroSim file is named “C:\pyrosim_files\switchgear.psm”, the results will be stored in “C:\pyrosim_files\switchgear\ ”. PyroSim will save a copy of the current PyroSim file into this directory, as well as the FDS input file, a Smokeview .ini file, and a .ge1 file containing detailed geometry. The input files will automatically be named after the PyroSim file. For the switchgear example, the files would be “switchgear.fds”, “switchgear.ini”, and “switchgear.ge1”. All result files from FDS will also be stored in this directory.

Next, the **FDS Simulation** dialog shown in Figure 18.10 is launched. This dialog, which shows FDS progress and messages, can be minimized and you can continue using PyroSim (and even run additional simulations) while a simulation is running.

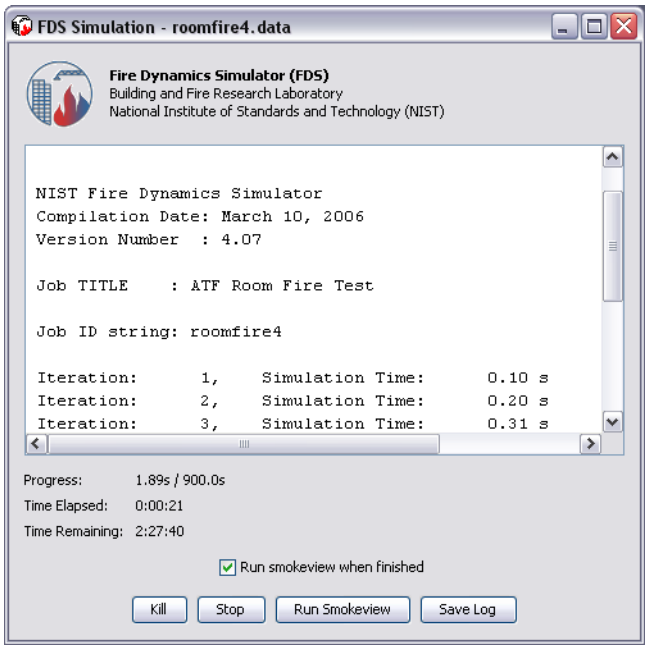

FDS Simulation - roomfire4.data
Fire Dynamics Simulator (FDS)
Building and Fire Research Laboratory
National Institute of Standards and Technology (NIST)
NIST Fire Dynamics Simulator
Compilation Date: March 10, 2006
Version Number : 4.07
Job TITLE : ATF Room Fire Test
Job ID string: roomfire4
Iteration: 1, Simulation Time: 0.10 s
Iteration: 2, Simulation Time: 0.20 s
Iteration: 3, Simulation Time: 0.31 s
Progress: 1.89s / 900.0s
Time Elapsed: 0:00:21
Time Remaining: 2:27:40
Run smokeview when finished
Kill
Stop
Run Smokeview
Save Log

Figure 18.10. The FDS simulation dialog

You can save the simulation log at any time by clicking **Save Log**. This log will be saved as a text file.

You can also run Smokeview while the simulation is in progress by clicking **Run Smokeview**. For details on how to use Smokeview, please consult the Smokeview User's Guide. Smokeview will run automatically when the simulation is finished.

Clicking **Stop** will cause PyroSim to create a .stop file that signals FDS to stop the simulation, but also write out a checkpoint file that can be used to resume the simulation later. There is often a significant delay between the time when you click the **Stop** button and when the simulation actually terminates. This is because FDS checks for the stop file at the same rate that it updates the progress information.

To immediately terminate the current simulation, you can click **Kill** or close the dialog. You will not be able to resume the current simulation.

NOTE: When starting a simulation or exporting an FDS file for some models, the user may receive the following message as shown in Figure 18.11: "PyroSim has detected a hole touching a mesh boundary, which may cause cutting problems in FDS. Would you like to slightly expand these types of holes?"

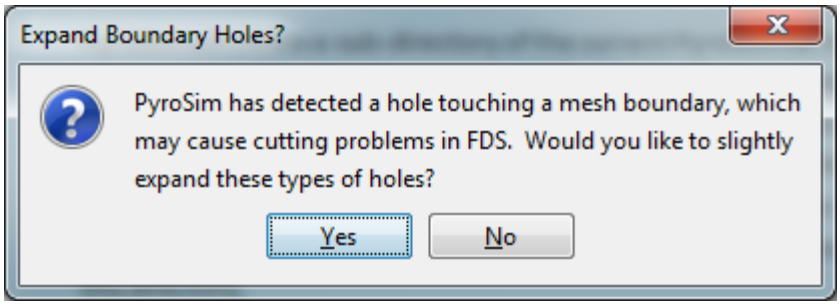

Expand Boundary Holes?
PyroSim has detected a hole touching a mesh boundary, which may cause cutting problems in FDS. Would you like to slightly expand these types of holes?
Yes
No

Figure 18.11. Expand Boundary Holes dialog

FDS currently has an issue where it will not fully cut a hole from an obstruction if both the hole and obstruction touch a mesh boundary at the same location. Instead, FDS leaves a thin obstruction along the mesh boundary. Figure 18.12 shows a model in PyroSim that can lead this problem. In this model, both the hole and the obstruction touch the bottom of the mesh, and the hole should cut all the way through the mesh. Figure 18.13 shows this model in FDS where the hole has not been punched all the way through the obstruction.

PyroSim detects potential cases where this might happen and prompts the user with the **Expand Boundary Holes** dialog. If the user chooses to expand the hole (the **Yes** option), PyroSim will expand the hole to 1/10 of a mesh cell past the mesh boundary for every side of the hole that touches a mesh boundary. This ensures the hole is properly cut all the way through the obstruction as shown in Figure 18.14. If the user chooses not to expand these types of holes (the **No** option), the hole will be written exactly as specified and may lead to the thin obstruction problem.

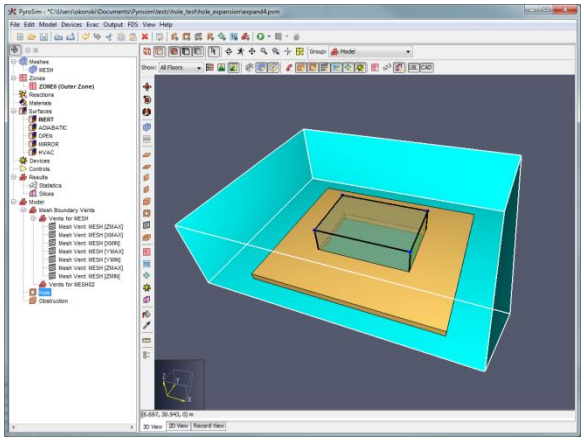


Figure 18.12. Hole along a mesh boundary

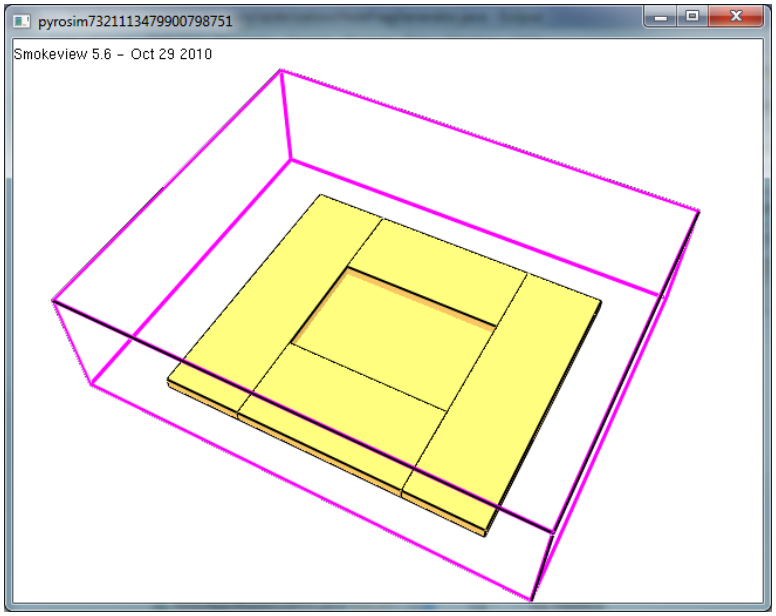


Figure 18.13. Improperly cut hole along mesh boundary in FDS

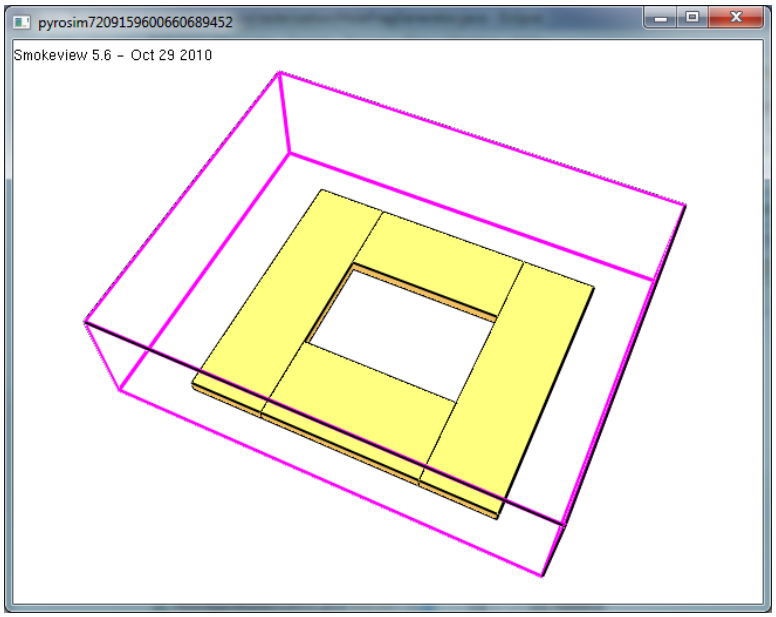


Figure 18.14. Properly cut hole along mesh boundary in FDS

Parallel Execution

PyroSim includes support to launch a parallel simulation using MPI. When running a simulation in parallel, all of the computation within each of the meshes can take place independently. Assuming a simulation executes in *t* seconds using only one processor, the best possible performance improvement

using *n* processors and *n* meshes is a reduction to *t/n* seconds[1]. In reality, this is not generally possible due to communications overhead and load balancing.

PyroSim's support for parallel execution is restricted to a single machine and is only beneficial on machines that have dual core or multiple CPUs.

To launch a parallel simulation in PyroSim, on the **FDS** menu, click **Run FDS Parallel...** or click the down arrow next to the **Run FDS...** button ( ) and select **Run FDS Parallel...**.

Before running a parallel simulation, you may want to take into account some guidelines:

- Use at least as many meshes as available processors (or cores). If there are 4 available processors and only two meshes, the additional two processors will not be used.
- Do not overlap meshes. Since information is exchanged between meshes at the edges, it is ideal to organize meshes in such a way that they touch, but do not overlap. It is not recommended to embed a fine mesh within a coarse mesh in an attempt to improve localized mesh resolution. Because information is exchanged at mesh boundaries, the outer mesh will not receive any data from the inner mesh.
- Do not allow a fire source to cross mesh boundaries. When a fire source crosses mesh boundaries, it is not possible to maintain the same level of simulation accuracy.

For a detailed list of suggestions and information about running FDS in parallel, please consult section 6.3.2 of the FDS Users Guide.

Cluster Execution

PyroSim supports the ability to run an FDS simulation on a network cluster using MPI. This has similar restrictions to running a parallel simulation, in that each grid is run in a separate process. The cluster may be composed of several computers, or nodes, and each node may have any number of processors.

Before launching a cluster simulation from within PyroSim, the node installer (available from the PyroSim website) must be installed on each computer that will participate, including the computer running PyroSim if it is to be part of the cluster. Running the node installer will install the MPI service and configure it appropriately for use by PyroSim.

If you have purchased the PyroSim cluster option, you will be able to run the simulation on any number of computers. If you have not purchased the cluster option, you will be limited to the computer running PyroSim plus one other.

To launch a cluster simulation within PyroSim, on the **FDS** menu, click **Run FDS Cluster...** or click the down arrow next to the **Run FDS...** button ( ) and select **Run FDS Cluster...**. This will launch the **Cluster FDS Parameters** dialog as shown in Figure 18.15.

[1] This is also referred to as a linear performance improvement, or linear speedup.

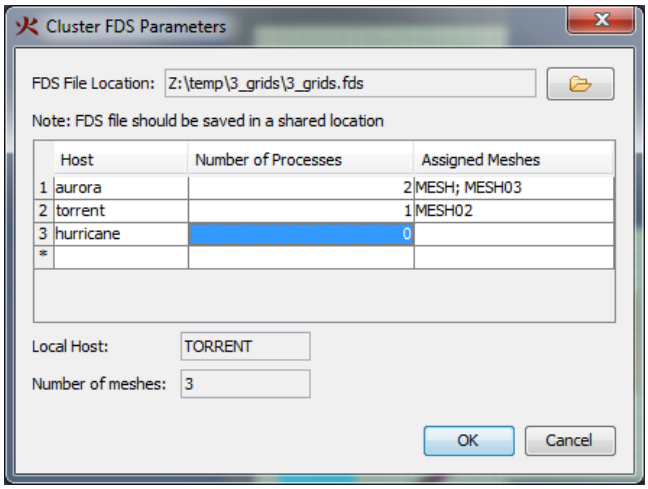


Figure 18.15. The Cluster FDS Parameters dialog

All nodes in the cluster can be entered in the table, along with the number of processes to launch on each node (NOTE: these nodes will be remembered the next time a cluster simulation is run). The **Assigned Meshes** column previews which FDS meshes will run on each computer, which is determined by the order of the meshes and the order of the hosts. The **FDS File Location** must be in a directory that is visible to all nodes in the cluster. Click the **OK** button to begin the simulation.

All input and output files will be stored in the same directory as the specified FDS file. In addition to the standard input files, PyroSim will also copy the appropriate FDS and MPI executables into the FDS file's directory. This ensures that all nodes in the cluster use the same versions of FDS and MPI.

Resuming a Simulation

If an FDS simulation has been gracefully stopped by pressing the **Stop** button in the simulation dialog, it can later be resumed. To do so, on the **FDS** menu, click **Resume Simulation...**. This will cause an additional `RESTART` flag to be written to the FDS input file. When FDS detects this flag it will automatically attempt to reload the previous execution state from the hard disk and resume where it left off. If FDS is unable to load the previous execution state, it will exit with an error.

Chapter 19. Post-Processing

PyroSim supports post-processing in two ways:

- Launching of Smokeview from within PyroSim, and
- Time history plots of output data.

Launching Smokeview

Smokeview is a post-processor for FDS supplied by NIST. It allows the user to view the FDS model along with results in 3D. The user can view animated smoke, slices, Plot3D, and various other output quantities.

By default, if you run FDS from within PyroSim, Smokeview will be launched at the end of the FDS run. Alternately, you can click on the FDS toolbar to launch the most recent results. You may also run Smokeview at any time by going to the **FDS** menu and selecting **Run Smokeview...**. This will prompt you to choose a Smokeview file to open.

Time History Results

Time history results are saved for heat detectors, thermocouples, and other fire output. After running an FDS simulation, the most recent plots can be viewed by clicking on the FDS toolbar. This will show a plot of thermal results. Device and control results may also be viewed by by clicking the down-arrow and selecting the desired plot. Additional plots may be shown by going to the **FDS** menu and choosing **Plot Time History Results...**. A typical heat detector plot is shown in Figure 19.1. The user can export the image to a file.

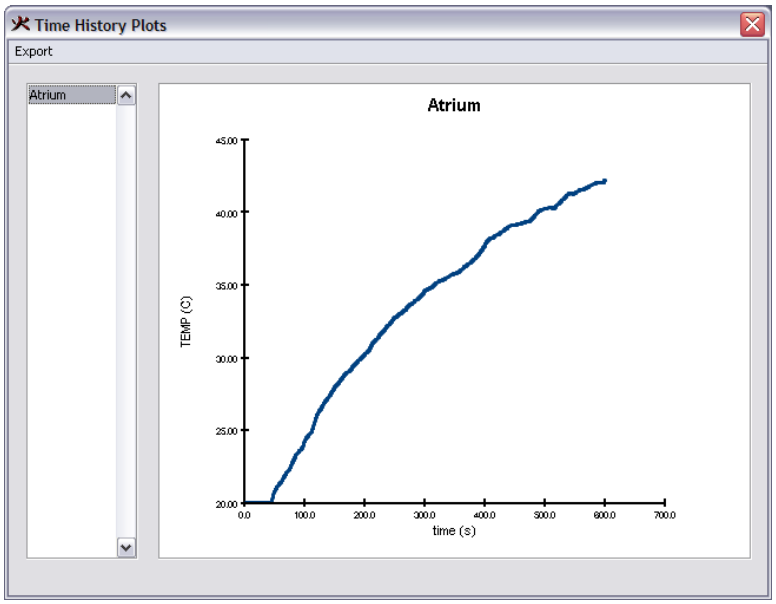


Figure 19.1. Time History Results

Archiving Results

After running a simulation, the results may be archived along with the FDS and PyroSim input files. To do so, on the **FDS** menu, click **Archive FDS Results...**. This will show the **Archive FDS Results** dialog as shown in Figure 19.2.

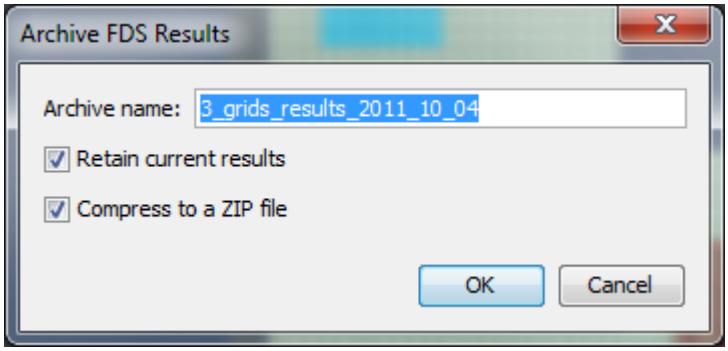


Figure 19.2. Archive FDS Results dialog

The **Archive name** identifies the archive for later retrieval. The default name contains the name of the model along with a date stamp. If **Retain current results** is checked, the current results will remain in place and a copy will be made for the archive. If unchecked, the current results will be deleted. If **Compress to a ZIP file** is checked, the archive will be stored in a compressed ZIP file. If unchecked, the archive will be stored as a duplicate folder of the current results. Press **OK** to create the archive. The archive will be stored in the directory of the current PyroSim file.

NOTE: In order for PyroSim to remember the archive for later retrieval, the PyroSim file must be saved after creating the archive.

Restoring Archived Results

Once results have been archived, they can be later restored. To do so, on the **FDS** menu, click **Restore FDS Results...**. This will show the **Restore Archived Results** dialog as shown in Figure 19.3.

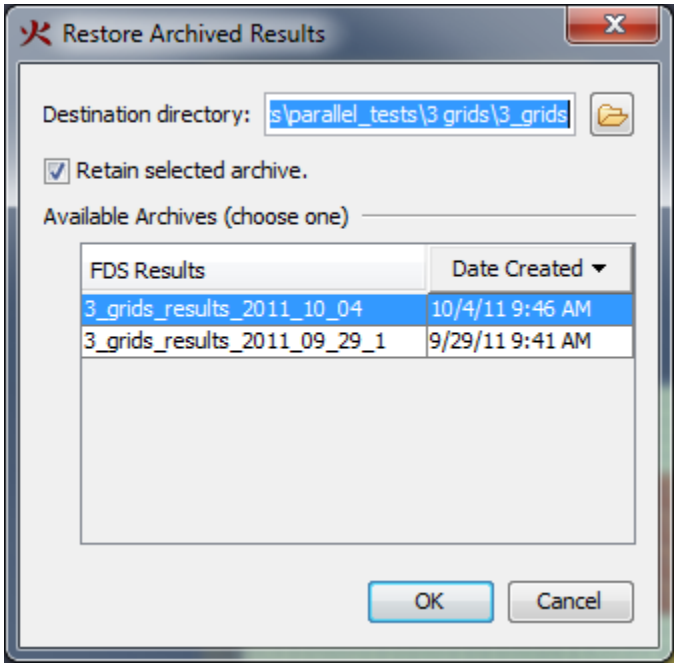


Figure 19.3. Restore Archived Results dialog

The **Destination directory** indicates the folder where the results will be restored. By default, this is the results directory of current PyroSim file. If **Retain selected archive** is checked, the selected archive will be kept on disk. If it is unchecked, the selected archive will be deleted. The table of **Available Archives** shows a record of all archives made under the current PyroSim file. Select one of these archives and then press the **OK** button to restore the archive.

Chapter 20. Managing Data Libraries

Libraries of material, or other model data, can reduce errors and speed the creation of new models. The user can import data from the library into a new model. This section describes how to manage PyroSim libraries.

Create and Manage Your Own Libraries

You can create and manage your own libraries for data that you commonly use. A library is a single file that can contain several categories of objects, such as Materials, Gas-phase Reactions, and Surfaces. To manage your library:

1. On the **Model** menu, click **Edit Libraries**.
2. Select the **Category** that you want to manage and move selected items from the **Current Model** into the **Library**, Figure 20.1.
3. Click **Save Current Library** and save the library in a location and with a name that you can access in the future.
4. Close the **PyroSim Libraries** dialog.

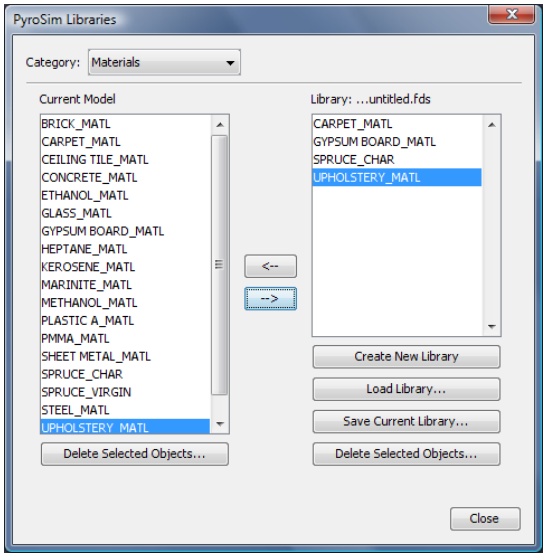


Figure 20.1. Creating a library of materials

After you have saved your library, you can load it into a new model and copy data from the library to your model.

Use the Library Provided with PyroSim

PyroSim includes a library of reaction and material data that has been gathered from the verification analyses provided with FDS. Each of these reactions and materials has a reference in the **Description** that documents the source of the data. This library is presently quite limited. NIST is supporting the

development of an engineering guide that will document the standard test methods used to obtain material properties.

To import data from the PyroSim database:

1. On the **Model** menu, click **Edit Libraries...**.
2. Click **Load Library** and open the `property library.fds` file that is found in the `C:\Program Files\PyroSim 2008\samples` folder.
3. In the **Category** box, select **Gas-phase Reactions** and copy appropriate reactions into your model.
4. In the **Category** box, select **Materials** and copy the appropriate materials into your model.
5. Close the **PyroSim Libraries** dialog.

Import a Material or Reaction from the FDS 4 Database

First, a caution. Version 4 of FDS provided a database that included several common materials and reactions. In version 5, the FDS developers made a conscious choice to remove material and reaction data. Many of the materials in FDS 4 were simply examples, and they were worried that users were applying them without using their own test or lab data as validation. In this section, we describe how to import the FDS 4 database, however, it is your responsibility to verify that this data is correct and applicable to your simulation.

You can convert the old FDS 4 materials and reactions for use in PyroSim 2007. To import FDS 4 data:

1. On the **File** menu, click **Import and select FDS File...**.
2. In the **Open** dialog, browse to `Program Files/PyroSim 2007/fds` folder and open the `database4.data` file.
3. You will receive conversion warning messages depending on what is converted and what properties are defined. Because of the change in material and reaction models between FDS 4 and FDS 5, you should review these carefully and edit the imported data appropriately for use with FDS 5. You can save the warnings to a file for future reference.
4. Click **OK** to close the **File Conversion Warnings** dialog.
5. The imported materials and reactions are now available for use in PyroSim 2007. **It is the user's responsibility to verify these values**.

Chapter 21. Advanced FDS Parameters

PyroSim tries to support FDS completely, but there are some more obscure features that might not be found in the PyroSim user interface. For these items, PyroSim provides additional mechanisms to allow these features through the **Additional Records** section of the **Record View** and the **Advanced** tab of some dialogs.

Additional Records Section

There are times when PyroSim does not support an entire record. In this case, the record can be entered into the **Additional Records Section** of the **Record View** as shown in Figure 21.1.

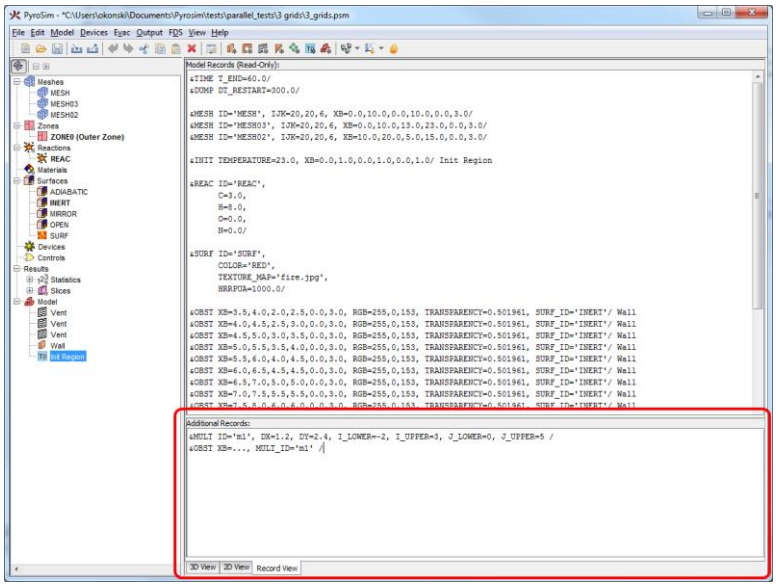


Figure 21.1. Additional Records Section

When PyroSim write the FDS input file, it copies the contents of the **Additional Records Section** exactly at the beginning of the FDS file.

Because PyroSim performs no validation on text in this view, it is up to the user to ensure that the statements are well-formed FDS statements and that they do not conflict with any records generated by PyroSim. In addition, none of the records written in this section can be referenced by other PyroSim objects. For instance, if a SURF record is entered in this section, it cannot be referenced by an obstruction in the PyroSim user interface. The only way to do so would be to write the obstruction in the **Additional Records Section** as well.

Some records, such as MISC, RADI, and others that occur once in the FDS input file can be entered in the **Additional Records Section** even if PyroSim has generated the record already. For instance, if PyroSim generates the record, "&MISC TMPA=30.0/", you can still enter another entry for MISC in the **Additional Records Section**, such as "&MISC P_INF=2.0E5/". FDS will merge these MISC records together to form "&MISC P_INF=2.0E5, TMPA=30.0/".

Advanced Parameters

Sometimes PyroSim may support a record but may not support it completely. For many of these records, including SURF, REAC, MATL, PART, TIME, DUMP, RADI, and MISC, PyroSim provides a way to enter additional fields. For Surfaces, Materials, Reactions, and Particles, there is an **Advanced** tab in the properties dialog where these additional fields may be entered, as shown in Figure 21.2. To enter additional fields for TIME, DUMP, RADI, and MISC, on the **FDS** menu choose **Simulation Parameters...**, then choose the **Misc.** tab.

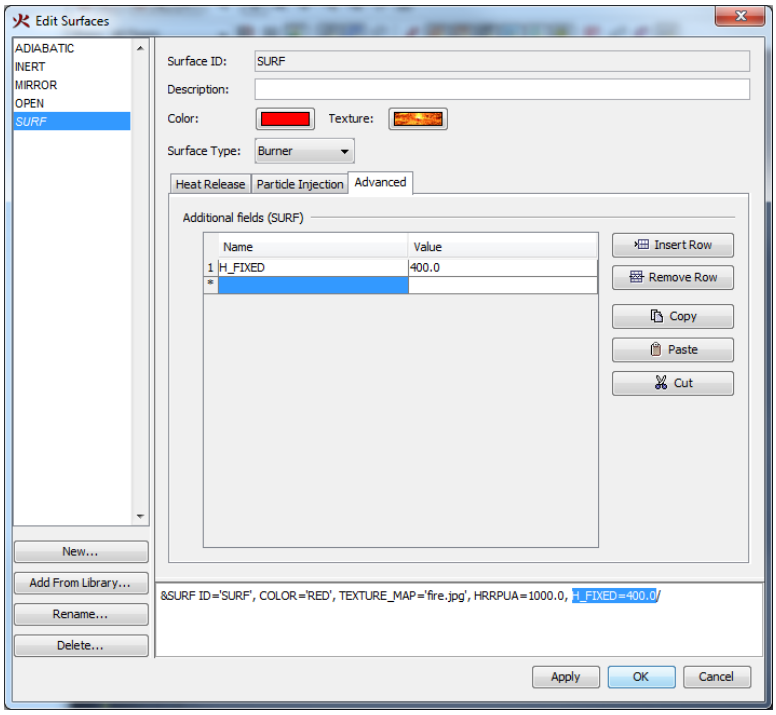


Figure 21.2. Additional Fields

When entering additional fields, you must specify the field name and the field value. These additional fields will then be appended to the FDS record generated by PyroSim. As in the **Additional Records Section**, PyroSim will write these fields to the file exactly as entered in the table, so care must be taken by the user to make sure they are correct.

Chapter 22. Troubleshooting

Licensing/Registration Problems

If you experience trouble registering PyroSim, please contact <support@thunderheadeng.com>.

Video Display Problems

PyroSim utilizes many advanced graphics card features in order to provide accelerated display of models in three dimensions. If you have problems with display, such as corruption of the image when you move the mouse, go to **File->Preferences** and turn off the fast hardware drawing options. This will disable the image caching and force PyroSim to always re-render the model. This should correct any display problems at the expense of speed.

You can also turn off graphics acceleration by starting PyroSim in Safe Mode. Select **Run**, **All Programs**, **PyroSim**, and then **PyroSim (Safe Mode)**.

If you encounter this problem, please let us know the make/model of your video card and what video driver you are using. That will help us improve the faster version to work on more computers.

Memory for Large Models

When running large models, it is possible that an out of memory error will be encountered. If this occurs, you can increase the default Java heap size. In our experience, the maximum size can be specified to approximately 70% of physical memory. By default, PyroSim will specify a java heap size of 50% of physical memory.

To specify the memory, you can either run from a command line or change the Start Menu shortcut properties. To run from a command line, open a command window and then go to the PyroSim installation directory (usually `C:\Program Files\PyroSim`). Execute PyroSim on the command line using the `-JXmx` argument. In this argument, the `J` specifies that the command will be passed along to the Java VM, not PyroSim. For example, `pyrosim -JXmx1200m` will request 1200 MB of memory.

To edit the PyroSim shortcut properties, right-click on the PyroSim icon, select the Shortcut tab, and then edit the Target by adding a space and `-JXmx1200` to the end of the Target. A typical Target will then read `"C:\Program Files\PyroSim\pyrosim.exe" -JXmx500m`.

MPI Problems

PyroSim Requires a Password to Run Parallel

MPI processes communicate using network protocols that are disabled by default for accounts without passwords. In order to work, MPI must have access to a password-protected account. Users without passwords can overcome this problem in a couple ways:

- Set a password for the account.

- Instruct MPI to authenticate using an alternate account.

To instruct MPI to authenticate using an alternate account (e.g. set to mpi_user on a computer named aurora), you must issue a command using the console.

```
Microsoft Windows [Version 6.0.6002]
Copyright (c) 2006 Microsoft Corporation.  All rights reserved.

C:\>cd "\Program Files\PyroSim 2013\fds"

C:\Program Files\PyroSim 2013\fds>mpiexec -remove
Account and password information removed from the Registry.

C:\Program Files\PyroSim 2013\fds>mpiexec -register
account (domain\ user) [aurora\thornton]: aurora\mpi_user
password:
confirm password:
Password encrypted into the Registry.
```

To verify that MPI will function with the account information:

```
C:\Program Files\PyroSim 2013\fds>mpiexec -validate -port 52700
SUCCESS
```

When trying to Run Parallel, PyroSim Repeatedly Requests a Password

PyroSim attempts to validate the MPI configuration prior to running the simulation. If this validation fails, PyroSim assumes it was because of a password mismatch. If you know this is not the case (e.g. you know you entered your password correctly), PyroSim may be responding incorrectly to a different error. To diagnose this error, please run PyroSim in safe mode. The error output should appear at the bottom of the console window. Forward this text to <support@thunderheadeng.com> and the support staff will help resolve the problem.

When Trying to Run Parallel, FDS Completes Immediately with No Output

This indicates that MPI started successfully, but the FDS executable (fds5_mpi) failed to run. To gather additional information about this error, you must run the MPI executable manually from the command prompt and observe the error output. To run the MPI executable manually, open a console window and issue the following commands:

```
Microsoft Windows [Version 6.0.6002]
Copyright (c) 2006 Microsoft Corporation.  All rights reserved.

C:\>cd "\Program Files\PyroSim 2013\fds"

C:\Program Files\PyroSim 2013\fds>fds5_mpi
```

The subsequent output should resemble the start of a successful FDS run; however, in this case it will probably contain error output. Copy this error output and email <support@thunderheadeng.com>, the support staff will help resolve the problem.

Contacting Technical Support

The PyroSim software is available for download at: http://www.thunderheadeng.com. The same site provides PyroSim user manuals and example problems. Please follow the examples to become familiar with the software.

Questions and suggestions should be sent to <support@thunderheadeng.com> or by phone to +1.785.770.8511.

Mail should be sent to:

```
Thunderhead Engineering
403 Poyntz Ave. Suite B
Manhattan, KS 66502-6081
USA
```

Appendix A. Opening FDS v4 and PyroSim v2006 Files

Due to the differences between versions 4 and 5 of FDS, it is not always possible to automatically convert legacy FDS input files and PyroSim 2006 PSM files to the new version. However, many conversions are possible and in many cases PyroSim can completely convert old input files to the new format.

PyroSim will begin the conversion process as a result of either of two actions: (1) opening a PSM file saved with a version of PyroSim designed to work with version 4 of FDS, and (2) importing an FDS input file designed to work with version 4 of FDS.

In many cases, PyroSim 2013 can import records intended for version 4 of FDS that PyroSim 2006 could not. This is because PyroSim 2013 supports a broader range of FDS features than the previous version. Examples of previously unsupported version 4 features that can now be imported include solid-phase thermocouples and extra species.

The process for converting PSM files and FDS input files is identical. PyroSim first loads the data into a form designed to work with version 4 of FDS, then applies conversion logic to produce the corresponding data structures designed to work with version 5 of FDS. For more information about how the data is converted from a format suitable for version 5 of FDS to version 6 of FDS, see Chapter 1.Appendix A. When PyroSim encounters a record that cannot be automatically converted, a warning message is generated. Each warning contains information about the source of the problematic record and the action taken. Some records are simply dropped and others are converted to default values. If a record is encountered that cannot be converted, but contained only default values and would not have affected the simulation, that record is dropped without issuing a warning.

Great care was taken to ensure that PyroSim generates these warnings whenever they contain important information, but not so often that they distract from important issues. When in question, PyroSim will err on the side of caution and generate a warning message. An example of this warning dialog is shown in Figure A.1. If no warning dialog appears, PyroSim was able to convert the input file without encountering any compatibility issues.

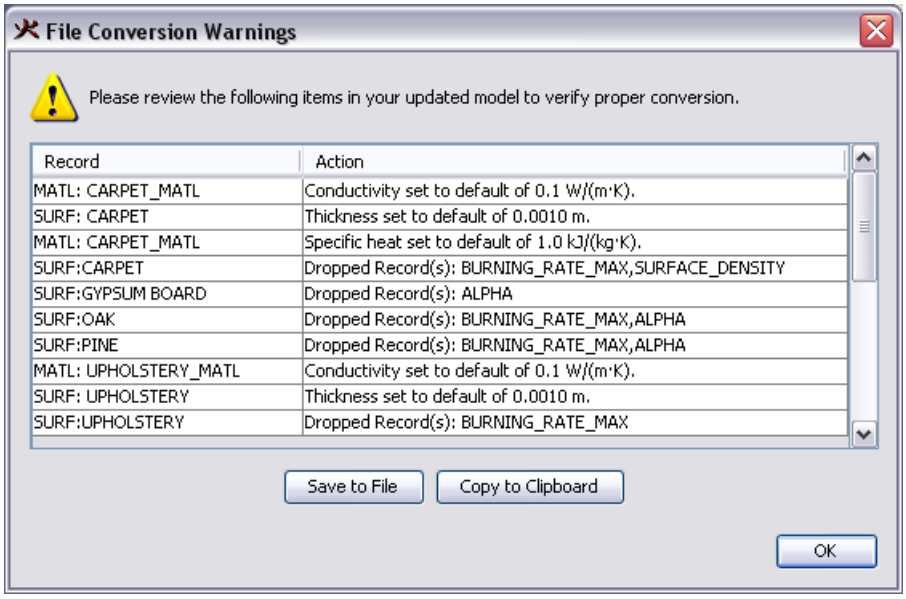


Figure A.1. Example import warnings

In most cases, the following records can be converted with no additional input:

- Geometry Data (walls, holes, triangles, etc...)
- Textures
- Mesh
- Floors
- Particles[1]
- Smoke Detectors
- Thermocouples
- Heat Detectors
- Boundary Quantity Output
- Plot3D Data
- Isosurfaces
- Slices
- Unsupported Records[2]

Global Simulation Parameters

The following items that can be set in the **Simulation Parameters** dialog of PyroSim 2006 are not supported in PyroSim 2013 and will be dropped.

- Under the **Simulator** tab, **Incompressible Calculation (excludes heat)**
- Under the **Environment** tab, **External Temperature**

[1] PyroSim 2006 did not properly support initial droplets or particle clouds. In PyroSim 2007 particle clouds are supported and existing particles with initial droplets specified will now be handled correctly.

[2] The unsupported records are copied verbatim from your previous version. Even though some of these records may now be supported, PyroSim will not perform any automated handling.

- Under the **Particles** tab, **Droplet Insert Interval**
- Under the **Particles** tab, **Max Particles per Second**

All other simulation parameters will be converted to PyroSim 2013 without warnings.

Note

In PyroSim 2013 it is possible to specify both the particle insertion interval and the particle insertion rate on a per particle basis. These options are available in the **Edit Particles** dialog, in the **Injection** tab. PyroSim does not automatically apply the global data to these fields.

Sprinklers and Pipes

All correctly specified sprinkler parameters are converted without warnings. If a sprinkler has been assigned a massless particle, however, that sprinkler will be assigned a particle with parameters from the make file, and a warning will be issued.

For FDS 4 sprinkler make files, PyroSim has a robust built-in parser that can handle both simple and complex spray patterns. The only requirement is that referenced make files must exist in the `fds` folder in the PyroSim install directory. PyroSim 2013 ships with the make files provided by NIST for FDS 4. If a file uses another make file, place it in this directory before importing or opening the file.

If there is a dry pipe delay greater than zero, PyroSim 2013 will create a single dry pipe with that delay and attach it to all the sprinklers in the model. Note, however, that in PyroSim 2013 the water pressure is specified per sprinkler rather than per pipe. Because of this, PyroSim will not convert the dry pipe pressure specified in the pipe record, and a warning will be issued.

Reactions

To convert reaction data into a form useable by version 5 of FDS, PyroSim 2013 must reverse-engineer the fuel molecule composition based on stoichiometric coefficients. To accomplish this, PyroSim uses the equations given in section 4.4.2 of the users guide for version 4 of FDS. The result is then checked to ensure that the total molecular weight is the same as the specified molecular weight. If this check succeeds, no warning will be issued. If the test fails, PyroSim will issue a “Converted stoichiometry” warning and you must manually update reaction data to ensure accurate simulation results.

Surfaces

Some surface properties are converted with no additional input or warnings, including surface names, colors, and textures. The different surface types, however, undergo more complicated conversions. The following describes how PyroSim 2006 surface types are converted to Surfaces and Materials in PyroSim 2013:

- Inert and Adiabatic - converted directly.
- Burner Fire - converted to a Burner Surface.
- Fan/Wind - converted to a Supply surface if the air flow is negative, an Exhaust surface, otherwise.
- Flammable Solid of Fixed Temperature or Heat Flux - converted to a Heater/Cooler surface.

- Thermally Thick/Thin Flammable Solid - converted to a Layered Surface with one single-step reacting Material.
- Flammable Solid (Constant HRR) of Fixed Temperature or Heat Flux - converted to a burner.
- Thermally Thick/Thin Flammable Solid (Constant HRR) - converted to a Layered Surface with one non-reacting Material. The reaction is controlled at the surface.
- Non-Flammable Solid of Fixed Temperature or Heat Flux - converted to a Heater/Cooler.
- Thermally Thick/Thin Non-Flammable Solid - converted to a Layered Surface with one non-reacting Material.
- Liquid Fuel - converted to a Layered Surface with one Liquid Fuel Material.
- Charring Fuel - converted to a Layered Surface with one layer. The layer is composed of a water and a virgin material. The virgin material undergoes one reaction where half of it is converted to fuel and the other half is converted to the charring material specified in the original surface. This ratio may need to be adjusted after conversion.
- Liquid Thermoplastic - converted to a Layered Surface with one Liquid Fuel Material.
- Charring Thermoplastic - converted to a Layered Surface with one single-step reacting Material.

Thermally Thin Surfaces

Unlike PyroSim 2006, PyroSim 2013 requires that every layered surface specify a thickness (Delta) for each layer and that materials specify density (Rho), specific heat, and conductivity (C). In PyroSim 2006, there were a number of ways for thermally thin surfaces to either specify or omit these parameters. These surfaces allowed any one or more of C, Delta, and Rho to be specified in addition to C*Delta*Rho. PyroSim 2013 will make a best-effort calculation of missing parameters. For instance, if C*Delta*Rho is specified along with two of the parameters, the third will be calculated; however, if more than one parameter is missing, PyroSim will use defaults for up to two of the parameters and calculate the third missing one. The default thickness for thermally thin surfaces is set to 1mm. In all cases where a default number has been assumed due to a missing parameter, a warning will be shown for the parameter.

Where is the Surface Database?

PyroSim 2013 does not currently ship with a surface database, but users can still make their own. In fact, many different objects can now be put into a database including materials and surfaces, extra species, reactions, particles, and several more. As common surface descriptions and other of these object properties become available from reliable sources in a format supported by version 5 of FDS, PyroSim will again ship with a pre-filled database.

Appendix B. Opening FDS v5 and PyroSim v2012 Files

Due to the differences between versions 5 and 6 of FDS, it is not always possible to automatically convert legacy FDS input files and PyroSim 2012 PSM files to the new version. However, many conversions are possible and in many cases PyroSim can completely convert old input files to the new format.

PyroSim will begin the conversion process as a result of either of two actions: (1) opening a PSM file saved with a version of PyroSim designed to work with version 5 of FDS, and (2) importing an FDS input file designed to work with version 5 of FDS.

The process for converting PSM files and FDS input files is identical. PyroSim first loads the data into a form designed to work with version 5 of FDS, then applies conversion logic to produce the corresponding data structures designed to work with version 6 of FDS. When PyroSim encounters a record that cannot be automatically converted, a warning message is generated. Each warning contains information about the source of the problematic record and the action taken. Some records are simply dropped and others are converted to default values. If a record is encountered that cannot be converted, but contained only default values and would not have affected the simulation, that record is dropped without issuing a warning.

Great care was taken to ensure that PyroSim generates these warnings whenever they contain important information, but not so often that they distract from important issues. When in question, PyroSim will err on the side of caution and generate a warning message. An example of this warning dialog is shown in Figure A.1. If no warning dialog appears, PyroSim was able to convert the input file without encountering any compatibility issues.

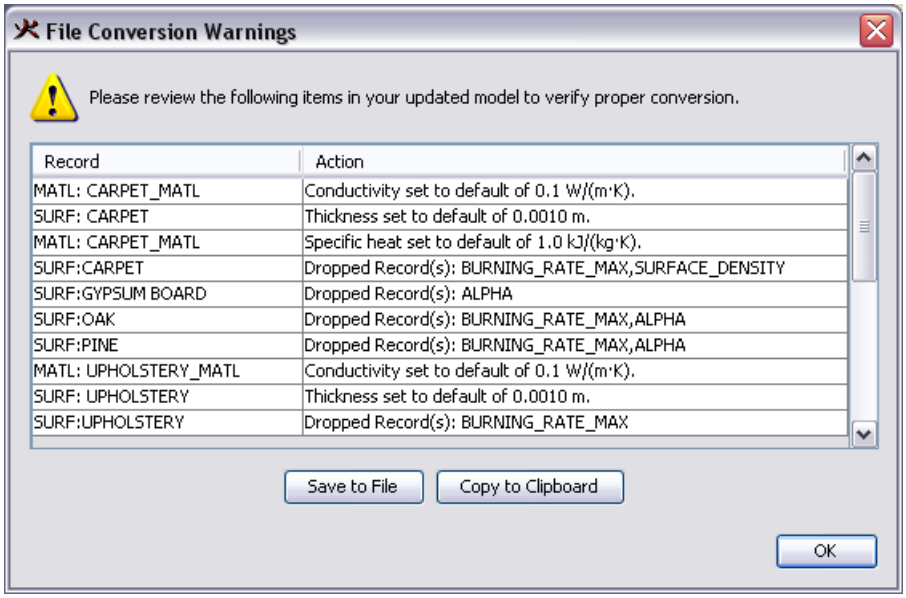


Figure B.1. Example import warnings

In most cases, the following records can be converted with no additional input:

- Geometry Data (walls, holes, triangles, etc...)
- Textures
- Mesh
- Floors
- Smoke Detectors
- Thermocouples
- Heat Detectors
- Boundary Quantity Output
- Plot3D Data
- Isosurfaces
- Slices
- Unsupported Records[1]

Global Simulation Parameters

The following items that can be set in the **Simulation Parameters** dialog of PyroSim 2012 are not supported in PyroSim 2013 and will be dropped.

- Under the **Simulator** tab, **Isothermal Calculation**
- Under the **Simulator** tab, **Include Gas Phase Flame Extinction**
- Under the **Simulator** tab, **Use Moinuddin and Li Turbulent Model**
- Under the **Simulator** tab, **Specify forced coefficient**
- Under the **Simulator** tab, **Specify horizontal coefficient**
- Under the **Simulator** tab, **Specify verticlal coefficient**
- Under the **Radiation** tab, **Use Additional Fuel Bands**

All other simulation parameters will be converted to PyroSim 2013 without warnings.

Reactions

While most reaction data can be converted easily, FDS6 does add new requirements to specify a valid reaction. Most notable is the requirement that a Fuel Species by specified by the user. This can be either a **Predefined** species, a **User Defined** species, or a **Default** species. When the **Default** species type is used, a generic, non-editable species called “REAC_FUEL” is added to the list of species. This species can be used the same way any other species would be, but its fields cannot be edited. All reactions defined in PyroSim 2012 or older automatically use the **Default** fuel type.

It should also be noted that in FDS6 based PyroSim versions, it is required that a reaction be active in order to simulate a fire. To make this transition easier, PyroSim automatically adds a reaction called “PROPANE_REAC” when converting PyroSim files that do not specify a reaction. The “PROPANE_REAC” attempts to mimic the default propane reaction that is used in FDS5.

[1] The unsupported records are copied verbatim from your previous version. Even though some of these records may now be supported, PyroSim will not perform any automated handling.

The following items will be dropped from the Reaction record:

- Under the **Fuel** tab, Mass Fraction of Fuel in Burner
- Under the **Fuel** tab**, Other atoms**
- Under the **Fuel** tab**, Molecular Weight**
- Under the **Fire Suppression** tab**, Limiting Oxygen Index**
- Under the **Heat Release Rate** tab, **Use Eddy Dissipation model for heat release rate**
- Under the **Heat Release Rate** tab**, C (C_EDC)**
- Under the **Byproducts** tab**, Hydrogen Fraction**
- Under the **Soot** tab**, Hydrogen Fraction**
- Under the **Soot** tab**, Maximum Visibility**
- Under the **Soot** tab**, Mass Extinction Coefficient**
- Under the **Fuel** tab, **Ambient Oxygen Mass Fraction** has moved to **Simulation Parameters**

Surfaces

Surfaces have undergone relatively few changes in from PyroSim 2012 to PyroSim 2013. However, a number of items are no longer supported in the new version. The following records will be dropped:

- A surface of type **Fan** will no longer set the FDS variable POROUS = .TRUE.
- Under the **Surface Props** tab, **Porous**
- Under the **Surface Props** tab, **Surface Density**
- Under the **Air Duct** tab, **Enable Air Duct**
- Under the **Air Duct** tab, **Duct Path**
- Under the **Air Duct** tab, **Maximum Over-pressure**
- Under the **Air Leak** tab**, Porous**

Particles

Unlike PyroSim 2006, PyroSim 2007 requires that every layered surface specify a thickness (Delta) for each layer and that materials specify density (Rho), specific heat, and conductivity (C). In PyroSim 2006, there were a number of ways for thermally thin surfaces to either specify or omit these parameters. These surfaces allowed any one or more of C, Delta, and Rho to be specified in addition to C*Delta*Rho. PyroSim 2007 will make a best-effort calculation of missing parameters. For instance, if C*Delta*Rho is specified along with two of the parameters, the third will be calculated; however, if more than one parameter is missing, PyroSim will use defaults for up to two of the parameters and calculate the third missing one. The default thickness for thermally thin surfaces is set to 1mm. In all cases where a default number has been assumed due to a missing parameter, a warning will be shown for the parameter.

In PyroSim 2013, the interaction between particles and species has changed significantly. In PyroSim 2012, a particle could be attributed various Thermal Properties and Fuel Properties. Most of these variables have since been moved to the species object.

To handle converting legacy files, PyroSim 2013 generates a new species based on the Thermal / Fuel Properties of the legacy particle. This species is then assigned under the Liquid tab of the PyroSim 2013 particle.

The following is a list of items which are applied to the generated species:

- Under the **Fuel** tab, **Melting Temperature**
- Under the **Fuel** tab, **Vaporization Temperature**
- Under the **Fuel** tab, **Heat of Vaporization**
- Under the **Fuel** tab, **Density**
- Under the **Fuel** tab, **Specific Heat**

Some items are not convertible. The following are dropped from the record:

- Under the **Fuel** tab, **Initial Temperature**
- Under the **Fuel** tab, **Heat of Combustion**

All other items are converted properly.

References

References

McGrattan, Kevin and Forney, Glenn. 2005. *Fire Dynamics Simulator (Version 4) User's Guide.* Washington, DC, USA : s.n., 2005. NIST Special Publication 1019.

McGrattan, Kevin. 2004. *Fire Dynamics Simulator (Version 4) Technical Reference Guide.* Washington, DC, USA : s.n., 2004. NIST Special Publication 1018.

McGrattan, Kevin, et al. 2007. *Fire Dynamics Simulator (Version 5) Technical Reference Guide.* Gaithersburg, Maryland, USA : s.n., October 2007. NIST Special Publication 1018-5.

McGrattan, Kevin, et al. 2007. *Fire Dynamics Simulator (Version 5) User's Guide.* Gaithersburg, Maryland, USA : s.n., July 2007. NIST Special Publication 1019-5.

McGrattan, Kevin, et al. 2013. *Fire Dynamics Simulator Technical Reference Guide Volume 1: Mathematical Model*. Gaithersburg, Maryland, USA : s.n. November 2013. NIST Special Publication 1018.

McGrattan, Kevin, et al. 2013. *Fire Dynamics Simulator User's Guide*. Gaithersburg, Maryland, USA : s.n., November 2013. NIST Special Publication 1019.